Seyi Olugbeja

The Ultimate Beginner's Guide to Professional CAD Design and Drafting Techniques

# AutoCAD 2025 For Beginners

# AutoCAD 2025

# For Beginners

## The Ultimate Beginner's Guide to Professional CAD Design and Drafting Techniques

*OceanofPDF.com*

Copyright © 2024 **Olugbeja Seyi**

**All Rights Reserved**

This book or parts thereof may not be reproduced in any form, stored in any retrieval system, or transmitted in any form by any means—electronic, mechanical, photocopy, recording, or otherwise—without prior written permission of the publisher, except as provided by United States of America copyright law and fair use.

**Disclaimer and Terms of Use**

The author and publisher of this book and the accompanying materials have used their best efforts in preparing this book. The author and publisher make no representation or warranties with respect to the accuracy, applicability, fitness, or completeness of the contents of this book. The information contained in this book is strictly for informational purposes. Therefore, if you wish to apply the ideas contained in this book, you are taking full responsibility for your actions.

*Printed in the United States of America*

*OceanofPDF.com*

# TABLE OF CONTENTS


**TABLE OF CONTENTS**

**INTRODUCTION**

**CHAPTER ONE**

**GETTING STARTED WITH AUTOCAD 2025**

- Notable Features of AutoCAD 2025
- Wide Range of Applications in Various Industries
- How to start using AutoCAD
- About AutoCAD 2025 Interface
  - *Quick Access Toolbar*
  - *Share Drawing*
  - *Application Menu*
  - *Ribbons*
  - *Command Window*
  - *InfoCenter*
  - *Graphical Area*
  - *Start Tab*
  - *Status Bar*
  - *What Is My Drawing Unit?*
- How to Create a New AutoCAD Drawing
- How to Open an Existing AutoCAD Drawing
- How to Save Your Drawings
- Essential Techniques for Saving in AutoCAD
  - *Using the Save command*
  - *Using the Save As command*
  - *Mastering Keyboard Shortcuts*
- Save Options and Settings
  - *AutoSave*
  - *Compatibility with different file formats*
  - *Backup files*
- Cloud Integration and Collaboration
  - *Saving to Autodesk Docs*
  - *Using the AutoCAD Web and Mobile Apps*

*Sharing Drawings*
Using the File Tab
*How to Close Drawing Files*
Using the Undo and Redo commands
*Undo command*
*Redo command*

**CHAPTER TWO**

**AUTOCAD DRAFTING**

What are the Drafting Priorities
How to Draw Lines Using the Line Command
Understanding Dynamic Input in AutoCAD 2025
*Exercises to Enhance Line Drawing Skills with Dynamic Input*
*Illustration of Specifying Lengths and Angles*
Exact Angles (Ortho vs. Polar Tracking)
*Increment Angle*
*Additional Angles*
*Polar Angle Measurement*
Precise Drafting using Object Snap
How to Enable Running OSNAPs
Using OSNAP Override
How to Draw Circles Using Circle Command
How to Draw Circular Arcs Using Arc Command
Different Object Snaps Linked to Circles and Arcs
How to Use Object Snap Tracking with OSNAP
How to Draw Lines and Ars Using Polyline Command
How to Convert Polylines to Lines and Arcs
*Joining lines and arcs to form a polyline*
How to Use Snap and Grid to Specify Precise Points
How to Use Polar Snap
CHAPTER DRAWING PROJECTS
*DRAWING LINES USING DYNAMIC INPUT*
*EXACT ANGLES PRACTICE*
*SNAP (OSNAP) Objective*
CONVERTING AND DRAWING POLYLINES
GRID AND SNAP

## CHAPTER THREE

## ADJUSTING COMMANDS

- Selecting Objects in AutoCAD 2025
  - *Select objects*
  - *Crossing Mode (C)*
  - *Window Mode (W)*
  - *Crossing Polygon Mode (CP)*
  - *Window Polygon Mode (WP)*
  - *Fence Mode (F)*
  - *Lasso Selection*
  - *Previous (P), Last (L), and more modes*
- How to Use Erase Command
- How to Use Selection Cycling
- How to Use Move Command
  - *How to Copy Command*
  - *How to Use Rotate Command*
  - *How to Use Scale Command*
  - *How to Use Strech Command*
  - *How to Use Mirror Command*
- How to Join Objects
- How to Lengthen Objects
- How to Use Grips and Dynamic Input
- How to Use Grips to Edit Objects

## CHAPTER FOUR

## ADJUSTING COMMANDS CONT'D

- About Offsetting Objects
  - *How to Offset Using the Through Option*
  - *How to Offset Using the Offset Distance Option*
- How to Use the Multiple Option
  - *How to Fillet Objects*
- How to Chamfer Objects
  - *Chamfering Using the Distance Option*
  - *Chamfering with Distance and Angle*
- How to Trim Objects

How to Extend Objects
Arraying Objects – Rectangular Array
How to Use the Array Creation Context Tab
*How to Edit a Rectangular Array Using Grips*
*How to Edit a Rectangular Array Using Quick Properties*
*How to Edit a Rectangular Array Using the Context Tab*
Arranging Objects – Polar Array
*Arranging Objects – Path Array*
How to Use Break Command

**CHAPTER FIVE**

**CIRCLES AND DRAWING AIDS**

How to Change the Grid Setting
*Selecting Options from the Command Line*
*How to Change the Snap Setting*
*Using the Drafting Settings Dialog Box*
How to Adjust Units
*The Drawing Units Dialog Box*
How to Draw Circles by Specifying a Center Point and a Diameter
*How to Draw Circles by Specifying a Center Point and a Radius*
Circle tool – drop-down menu
*How to Use Single-Point Object Snap*
*Use object snap to easily locate the quadrant point.*
*Locate the tangent point using object snap*
How to Use ERASE Command
*Verb/Noun Editing*
*Noun/Verb Editing*
How to Customize Your Workspace
*Saving the existing workspace*
How to Use RECTANG Command
How to Plot or Print a Drawing
Setting up a Printer
*Plot Area*
*Plot Preview*
*Important*
Chapter Drawing Projects

**CHAPTER SIX**

**COPIES, TEMPLATES, AND ARRAYS**

How to Set Drawing Limits
*Model Space and Paper Space*
How to Create and Save a Drawing Template
*NOTE*
How to Save a Drawing Template
*Note*
*Take note*
How to Use the Copy Command
*Copying with Grips*
How to Use the MOVE Command
Moving with Grips
*Moving by Typing a Displacement*
How to Use the ARRAYRECT Command
The Array Creation Contextual Tab
*Note*
*Important*
Using Multifunctional Grips
How to Adjust Plot Settings
*The Printer/Plotter Panel*
*Add-a-Plotter Wizard*
*Paper Size*
Understanding Drawing Orientation
*Plot Area*
*Plot Offset*
How to Create Center Marks
Chapter Drawing Projects
*Important*

**CHAPTER SEVEN**

**UNDERSTANDING DIMENSIONS**

How to Create and Save a Dimension Style
Creating Linear Dimensions in AutoCAD
How to Draw Multiple Linear Dimensions Using QDIM

*Important*
*Important*
Using DIMBASELINE and DIMCONTINUE
*Important*
Drawing Ordinate Dimensions
*Understanding QDIM and the Datum Point Option*
*QDIM prompts*
Understanding DIMORDINATE and the UCS Command
*AutoCAD prompts*
*AutoCAD prompts*
How to Draw Angular Dimensions
Angular Dimensions on Arcs and Circles
*About Dimensioning Arcs and Circles*
*Diameter Dimensions*
Radius Dimensions
*Create a radius dimension on the smaller circle*
How to Annotate with multileader
The Leaders Tool Palette
How to Change Dimension Text
*Using the DIMEDIT command*
The Quick Properties and Properties Palettes
How to Use the HATCH Command
*AutoCAD prompts*
*Note*
*Additional Predefined Patterns*
Gradient Fill
Scaling Dimensions Between Paper Space and Model Space
*Inserting text in paper space*
*Text in Paper Space and Model Space*
The Annotative Property
*Controlling Annotation Scale Visibility*
How to Use Associative Dimensions
Changing Associativity of Dimensions
Chapter Drawing Projects
*Drawing 8-1: Tool Block [INTERMEDIATE]*
*Drawing 8-2: Flanged Wheel [INTERMEDIATE]*

*Drawing 8-3: Knob [INTERMEDIATE]*
*Drawing 8-4: Nose Adapter [INTERMEDIATE]*
*Drawing 8-5: Plot Plan [INTERMEDIATE]*
*Drawing 8-6: Panel [ADVANCED]*
*Drawing 8-7: Angle Support [ADVANCED]*

**CHAPTER EIGHT**

**BLOCKS AND HATCH**

What is the Purpose of Blocks?
*Creating a block in AutoCAD*
How to Use (Insert) Blocks
Scale
*Auto-Placement*
*Repeat Placement*
Block Insertion Point OSNAP
Block Count
*Block Replace*
How to Explode Blocks and Convert Them to Files
*Exploding Blocks*
*Converting Blocks to Files*
How to Hatch in AutoCAD 2025
*Using the Hatch Command*
*How to Control Hatch Properties*
*Important*
*How to Control Hatch Options*
*How to Specify Hatch Origin*
*Specify the origin point*
Creating Associative Hatching
*How to Hatch an Open Area*
How to Use Match Properties to Create Similar Hatches
How to Create Separate Hatches in the Same Command
*How to Set Hatch Draw Order*

**CHAPTER NINE**

**TEXT**

How to Enter Single-Line Text with Justification Options

*Right-Justified Text*
*Centered Text*
MIDDLE TEXT
*Aligned Text*
*Text Drawn to Fit Between Two Points*
HOW TO ENTER MULTILINE TEXT USING MTEXT
HOW TO ENTER TEXT ON AN ANGLE AND TEXT USING CHARACTER CODES
THE DEGREE SYMBOL AND OTHER SPECIAL CHARACTERS
HOW TO EDIT TEXT IN PLACE WITH TEXTEDIT
HOW TO USE SPELL AND FIND COMMANDS
HOW TO EDIT TEXT WITH THE QUICK PROPERTIES PALETTE
ALIGNING MULTILINE TEXT IN AUTOCAD
HOW TO CHANGE PROPERTIES WITH MATCHPROP
*Align and slant vertical text*
HOW TO CHANGE STYLES AND FONTS
*Expanded Ribbon Panels*
*NOTE*
SWITCHING THE CURRENT STYLE
HOW TO CREATE TABLES AND FIELDS
*Table tool*
ADDING FIELDS
USE THE INSERT FIELD TOOL
HOW TO SCALE PREVIOUSLY DRAWN ENTRIES
*The scale tool*
SCALING BY REFERENCE
*Scaling with Grips*
HOW TO USE DRAWING TEMPLATES, TITLE BLOCKS, AND BORDERS
*Inserting text*
CHAPTER DRAWING PROJECTS
*Drawing 7-1: Title Block [INTERMEDIATE]*
*Drawing: Gauges [INTERMEDIATE]*
*Drawing: Stamping [INTERMEDIATE]*
GRIP COPY MODE WITH OFFSET SNAP LOCATIONS
*Drawing: Control Panel [INTERMEDIATE]*
ADJUSTING THE ORIGIN USING THE UCS COMMAND
*Drawing: Angle Bracket [ADVANCED]*

**CHAPTER TEN**

**PLOTTING**

Introduction to Paper Space and Model Space
What is Layouts?
Creating New Layout from the Beginning
Creating a New Layout Using a Template
*How to Create Layouts Using Copying*
How to Create Viewports
*Adding Single Rectangular Viewports*
How to Add Multiple Rectangular Viewports
*Adding polygonal viewports*
*Creating Viewports by Converting Existing Objects*
*Creating Viewports by Clipping Existing Viewports*
*Managing Viewports After Creation*
How to Freeze Layers in Viewports
How to Scale and Maximize Viewports
*First approach*
*Second approach*
How to Laver Override in Viewports
How to Use Plot Command

**2D PRACTICE DRAWINGS**

**INDEX**

*OceanofPDF.com*

# INTRODUCTION

This newest version of computer-aided design (CAD) software, AutoCAD 2025, was created by Autodesk and represents the most recent breakthrough in the field. AutoCAD, which was first made available to the public in 1982, has been a fundamental tool in the disciplines of architecture, engineering, and construction ever since it was first introduced. It is renowned for its accuracy and adaptability. This heritage is carried on by AutoCAD 2025, which introduces a series of new features and advancements with the goals of enhancing the user experience, increasing productivity, and allowing more complex design capabilities. Through the implementation of expanded 3D modeling capabilities, greater collaboration and cloud integration, and advanced automation possibilities, AutoCAD 2025 is well-positioned to fulfill the ever-evolving requirements of professionals working in a wide range of sectors, therefore establishing a new benchmark in the realm of design and drafting technology.

*OceanofPDF.com*

# CHAPTER ONE
# GETTING STARTED WITH AUTOCAD 2025

AutoCAD 2025, created by Autodesk, represents the newest version of a highly popular computer-aided design (CAD) software used globally. AutoCAD has completely transformed the AEC industry since it was first introduced in 1982. This software has become an essential tool for architects, engineers, and construction professionals, providing them with an incredibly robust platform for drafting, modeling, and visualizing their projects. Autodesk continuously improves AutoCAD by incorporating user feedback and leveraging technological advancements. This results in more intuitive, efficient, and versatile software with each new release. AutoCAD 2025 builds upon its legacy by introducing a wide range of innovative features and enhancements that address the ever-changing requirements of its diverse user community.

To truly grasp the significance of AutoCAD 2025, it is essential to take a glance at the software's evolution. AutoCAD was first introduced as a desktop application designed to run on microcomputers equipped with internal graphics controllers. AutoCAD revolutionized the field of drafting by eliminating the need for manual techniques and empowering drafters to produce designs with unparalleled accuracy and efficiency. Throughout its

evolution, AutoCAD has undergone numerous updates by Autodesk, incorporating a range of advanced features including 3D modeling, parametric constraints, and cloud-based collaboration tools. Every version of AutoCAD has built upon its predecessor, seamlessly integrating the latest advancements in computing power and user interface design. The transition from 2D drafting to 3D modeling marked a significant milestone in the field, empowering designers to craft intricate and comprehensive depictions of their projects. The AutoCAD web and mobile apps have revolutionized the way users work by offering unparalleled flexibility to work from any location.

# Notable Features of AutoCAD 2025

AutoCAD 2025 brings forth a multitude of fresh features and enhancements that are meticulously crafted to optimize workflows and boost productivity. **These are some of the notable additions:**

1. **Improved Performance and Stability:**

- **Optimized Rendering Engine:** The rendering engine in AutoCAD 2025 has been enhanced to deliver faster and more precise visualizations of 3D models. You will experience noticeable enhancements in rendering times, enabling faster iterations and more intricate presentations.
- **Enhanced File Handling:** AutoCAD 2025 now offers improved file handling capabilities, allowing for smoother navigation and editing of large and intricate drawings. Users will benefit from improved stability and enhanced performance, resulting in a more efficient working environment.

2. **Advanced 3D Modeling Tools:**

- **Parametric Design Improvements:** AutoCAD 2025 introduces enhanced parametric design tools, providing users with even more powerful and versatile options for creating and modifying 3D models. With the latest update, you can establish intricate connections between various components of their designs. This

empowers them to create more sophisticated and adaptable models.

- **Advanced Mesh Modeling:** The mesh modeling tools have undergone significant enhancements, enabling users to achieve greater precision and intricate control over intricate surfaces. This feature is especially valuable for designers who are working on complex geometries and organic shapes.

3. **Cloud Integration and Collaboration:**

- **AutoCAD Web App Improvements:** The web version of AutoCAD has undergone substantial updates, which have greatly enhanced its functionality and brought it closer to the desktop version. With the latest updates, users can seamlessly execute complex tasks right from their web browsers, greatly enhancing the practicality of remote work.
- **Advanced DWG Compare:** The DWG compare feature has been enhanced to provide users with more intuitive and detailed comparisons between different versions of drawings. This feature greatly enhances team productivity by facilitating the identification of changes and revisions, thereby fostering seamless collaboration and minimizing the occurrence of errors.

4. **Customization and User Interface:**

- **Streamlined Ribbon Interface:** The user interface in AutoCAD has been enhanced for improved usability. The ribbon interface has been streamlined and now offers customizable tabs and panels, allowing users to personalize their workspace to match their specific workflows. You can access the necessary tools with enhanced speed and efficiency.
- **Dark Mode Advancements:** Dark mode has been optimized to provide improved contrast and readability, resulting in reduced eye strain during extended work sessions.

5. **Scripting and Automation:**

- **Improved LISP and Python Support**: AutoCAD 2025 has expanded its support for automation by enhancing its LISP and Python scripting capabilities. By using this feature, users can streamline their work processes and enhance efficiency by automating repetitive tasks and designing personalized workflows.
- **New Machine Learning Tools**: Introducing new machine learning tools in AutoCAD 2025 that leverage advancements in artificial intelligence. These tools can assist with tasks like predictive text input and error detection in designs.

6. **Enhancing Visuals and Delivering Presentations:**

- **Textures and Realistic Materials:** The materials library has been significantly expanded and refined, providing a wide range of realistic and high-quality textures that can be used to enhance the quality of renderings. Mastering this skill is crucial for producing realistic visualizations that will leave a lasting impression on clients and stakeholders.
- **Enhanced Annotation Tools**: The latest version of AutoCAD offers upgraded annotation tools that offer increased flexibility and precision when it comes to adding notes, dimensions, and other crucial information to your drawings.

# Wide Range of Applications in Various Industries

**AutoCAD 2025 is an incredibly versatile tool that is used in various industries. Several industries greatly benefit from the advanced features of AutoCAD:**

- **Architecture:** AutoCAD is a powerful tool that architects rely on to create highly detailed floor plans, elevations, and sections. With its exceptional precision and advanced capabilities in handling intricate geometries, the software is perfectly suited for designing a wide range of structures, including residential and commercial buildings. With the advanced 3D modeling tools in AutoCAD 2025, architects can now bring their designs to life,

providing a more immersive and captivating experience for their clients.

- **Engineering**: AutoCAD is widely recognized and used in the engineering industry. It serves as a powerful tool for creating intricate designs of mechanical components, electrical systems, and various infrastructure projects. The new parametric design features in AutoCAD 2025 are incredibly valuable for engineers, allowing them to effortlessly create precise and flexible models. The enhanced file-handling capabilities greatly simplify the process of working with extensive and intricate engineering drawings.
- **Construction**: AutoCAD is widely used in the construction industry to generate precise construction documents and plans. The software's collaboration and cloud integration features enhance communication and coordination among project teams, ensuring seamless collaboration. The DWG compare tool in AutoCAD 2025 is a valuable resource for construction professionals, allowing them to easily track revisions and ensure that all team members are working with the latest plans.
- **Manufacturing**: AutoCAD is widely used by manufacturers to design various products, machinery, and production lines. The enhanced 3D modeling tools and upgraded mesh modeling capabilities in AutoCAD 2025 are extremely advantageous when it comes to crafting intricate and precise product representations. The automation and scripting features are incredibly useful for streamlining repetitive tasks and boosting efficiency.
- **Interior Design**: AutoCAD is a powerful tool that interior designers rely on to meticulously craft layouts and bring spaces to life through visualization. With the advanced materials and textures available in AutoCAD 2025, designers can now bring their creations to life with stunning realism. This assists clients in gaining a comprehensive understanding of the proposed layouts, enabling them to make well-informed decisions.
- **Urban Planning**: AutoCAD is a powerful tool that urban planners rely on to create intricate maps and meticulously plan the layouts of cities and towns. The enhanced annotation tools in AutoCAD 2025 enable planners to incorporate highly detailed

and accurate information into their maps, thereby enhancing communication and facilitating more informed decision-making.

# How to start using AutoCAD

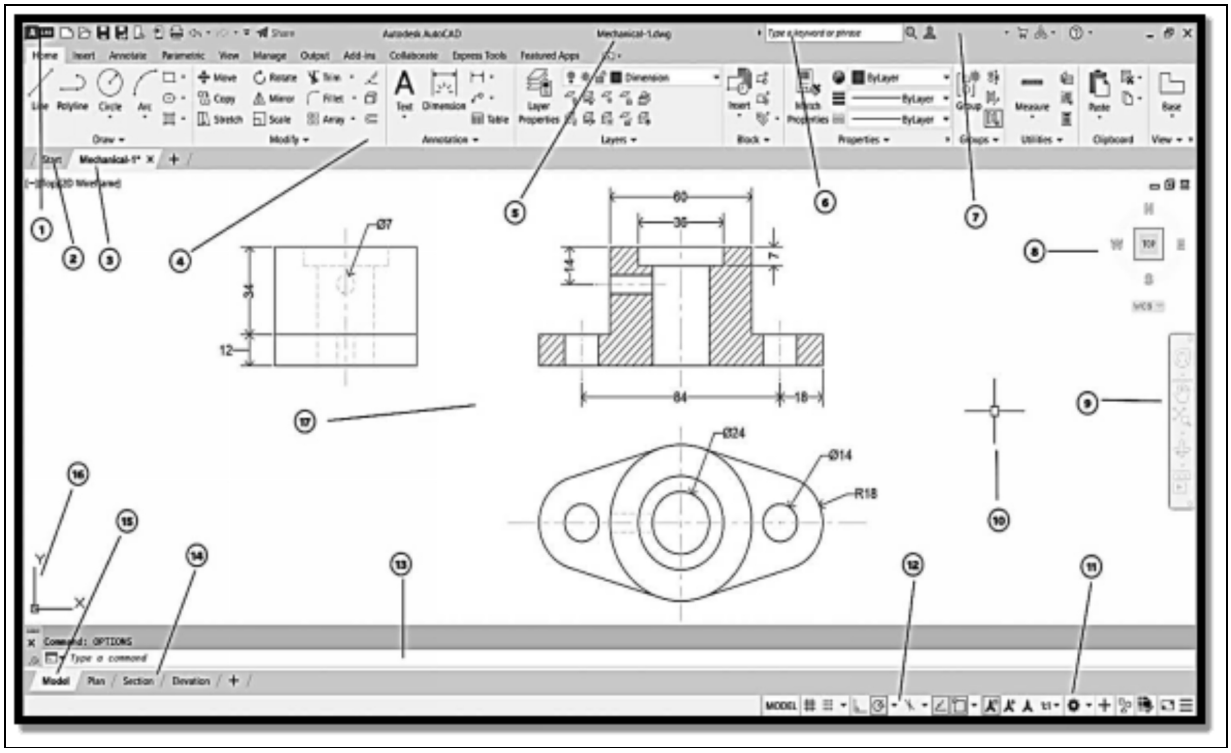
AutoCAD was first introduced in 1982 by Autodesk, Inc. when the company was still in its early stages. It was specifically designed for use on personal computers. AutoCAD has consistently maintained the largest user base in the CAD industry, solidifying its position as a global leader. AutoCAD is a versatile software that allows users to effortlessly create both 2D and 3D drafts and designs. AutoCAD is a versatile software that caters to a wide range of industries, including architecture, engineering, and manufacturing. It is commonly used for creating detailed drawings and designs for various purposes, such as buildings, roads, and machinery. AutoCAD remains the top choice for Autodesk, Inc. as it continues to be the most profitable software. Its user-friendly interface and comprehensive features cater to all user requirements, even in the era of BIM (Building Information Modeling). AutoCAD LT is a specialized version of AutoCAD that focuses solely on 2D drafting. To launch AutoCAD 2025, simply double-click on the shortcut icon located on your desktop, which was automatically generated during the installation. The Welcome window will be displayed by AutoCAD, as depicted below:

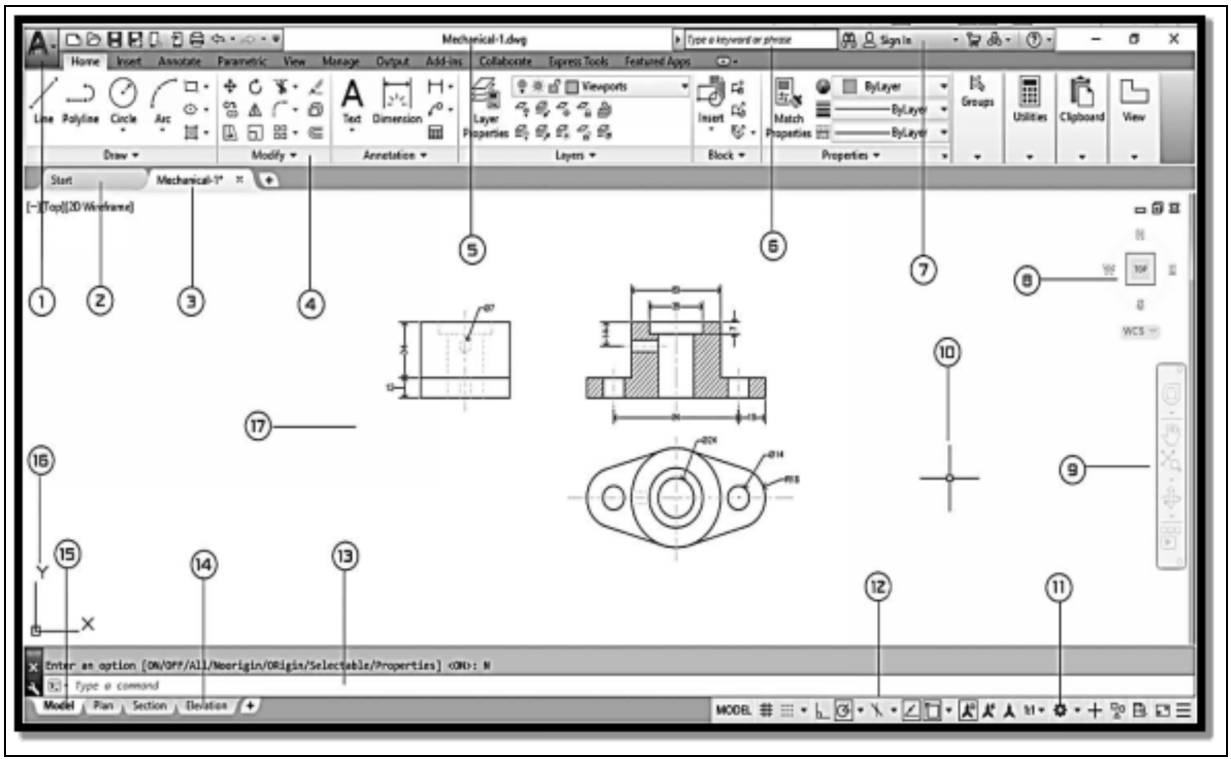
**When you're in the welcome window, you can:**

- Access an existing file, Open a Sheet Set, and Familiarize yourself with the sample files provided by the software.

- Begin a new drawing or access additional templates online.
- View your recently opened files either as a list or as icons.
- Make sure to check if there are any announcements from AutoCAD regarding your software or hardware.
- Direct any feedback you may have to Autodesk.

**When you begin a new project or access a previously created file, the AutoCAD 2025 interface will be displayed. Here is a visual representation:**





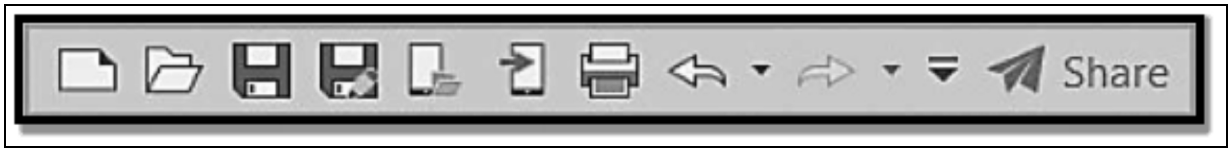

1. Application Menu
2. Start tab
3. File tab
4. Ribbon
5. Current file name
6. Info center
7. Sign in
8. ViewCube
9. Navigation bar
10. Crosshairs
11. Workspace
12. Status bar
13. Command window
14. Layout tab
15. Model tab
16. UCS icon
17. Graphical area

# About AutoCAD 2025 Interface

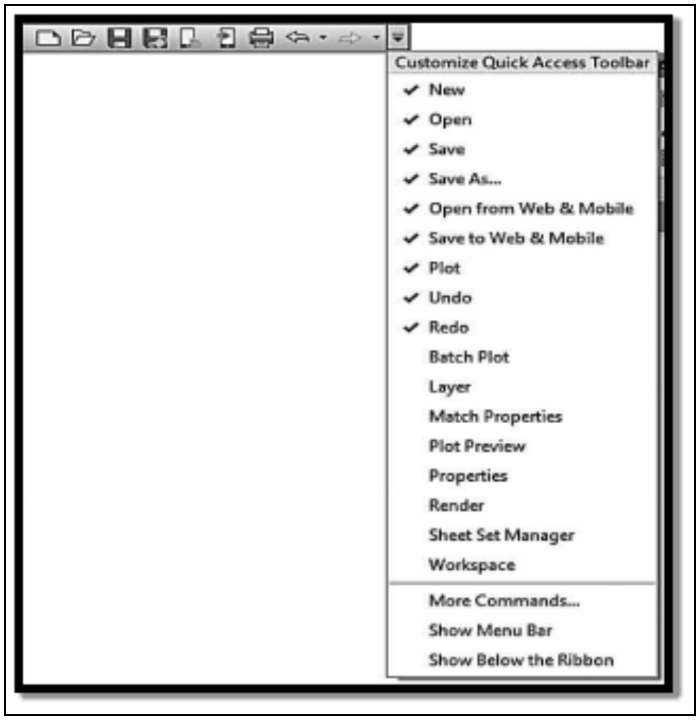
The interface of AutoCAD is built around Ribbons and an Application Menu. One of the key advantages of this interface is the increased size of the Graphical Area.

# Quick Access Toolbar

This toolbar includes all the File commands mentioned in the Application Menu, as well as **Workspace** and **Undo/Redo options**.



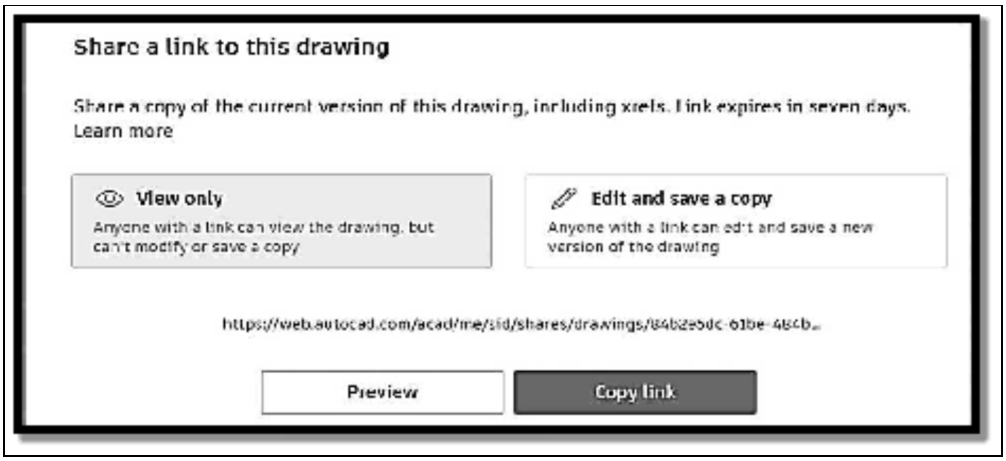

**To customize this toolbar, simply click the arrow at the end. This action will bring up the following options:**




As observed, you can add or remove commands. It may also be beneficial to click on "**Show Menu Bar**" as the ribbon does not encompass all of the AutoCAD commands.

# Share Drawing

When you use this command, AutoCAD Web will provide you with a message that includes a link to the current version of the drawing, along with its external references.




It is possible to create a link that allows others to view the content or even edit and save their copy.

# Application Menu

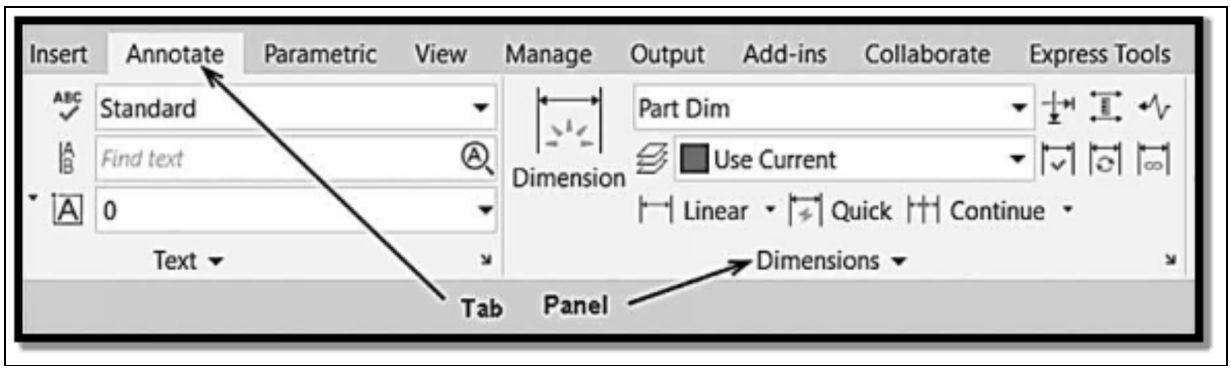
**The Application menu houses a variety of commands that are related to files:**

Commands for various file operations are available in AutoCAD. These include creating a new file, opening an existing file, saving the current file, saving the current file under a new name and in a different folder, importing from different file formats, exporting the current file to a different file format, printing and publishing the current file, and more. We will cover nearly all of these commands in various sections throughout this book. By default, the recent files will be displayed. To customize the display of recent files in the Application Menu, utilize this control:

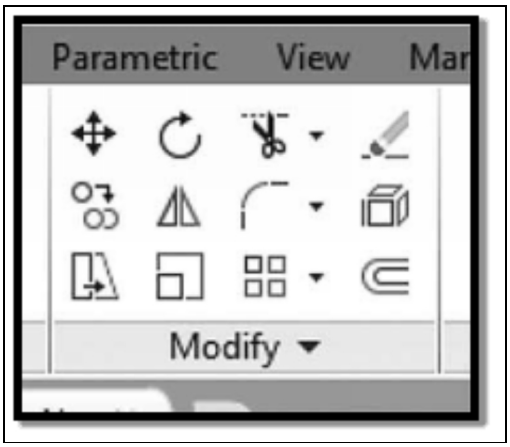
In addition, you have the option to customize the sorting of your recent files using this control:

# Ribbons

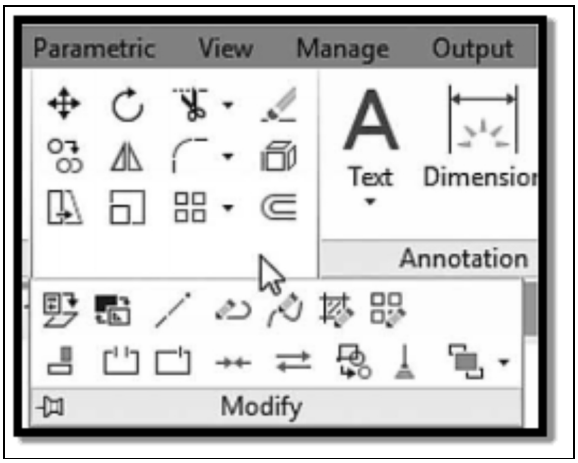
**Ribbons are composed of two main components: tabs and panels, as illustrated in the diagram below:**



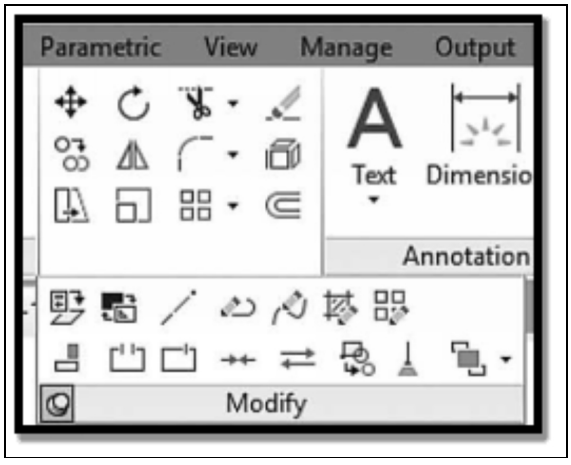

The Modify panel can contain additional buttons that are not currently visible.



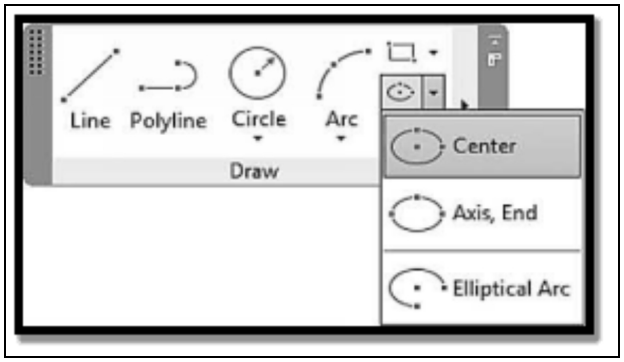

To access the additional options, simply click on the small triangle located next to the title.



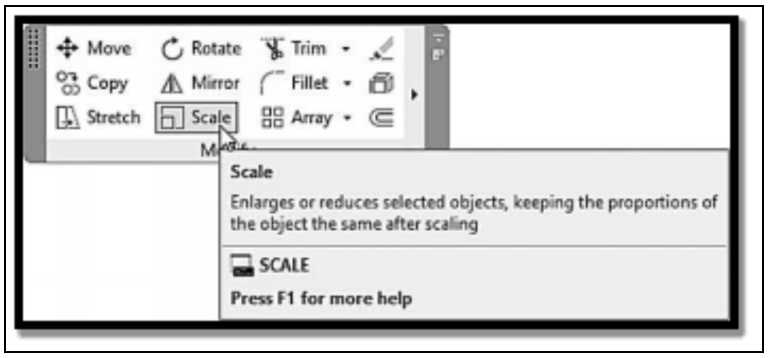

When you distance yourself from the panel, the buttons will vanish. To ensure constant visibility, simply click on the push pin. Here are the new views:



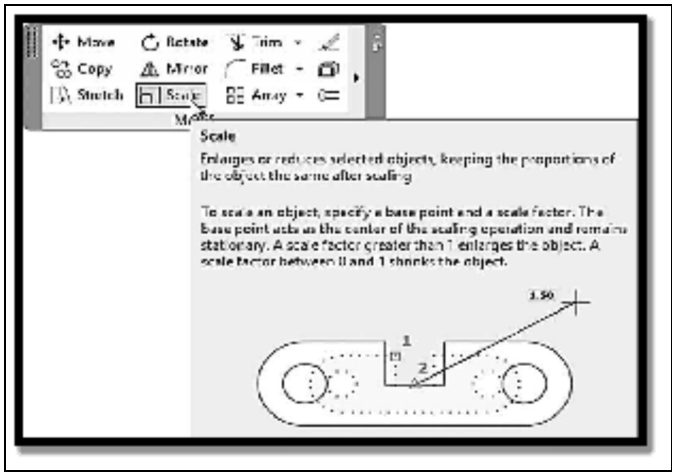
There are numerous options available for certain commands. For the convenience of users, AutoCAD has consolidated all the options into a single button. Take a look at the illustration below:



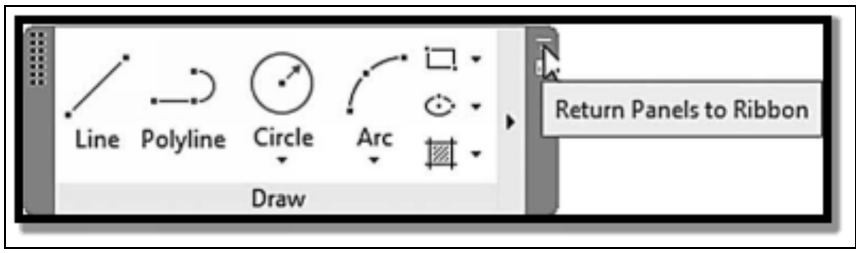

The ribbons offer a straightforward help feature. When you hover your mouse over a button, a small help screen will appear.



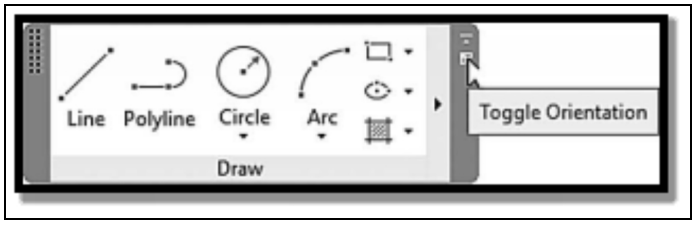

When the user hovers the mouse over the button for an extended period, AutoCAD will display more comprehensive help information.



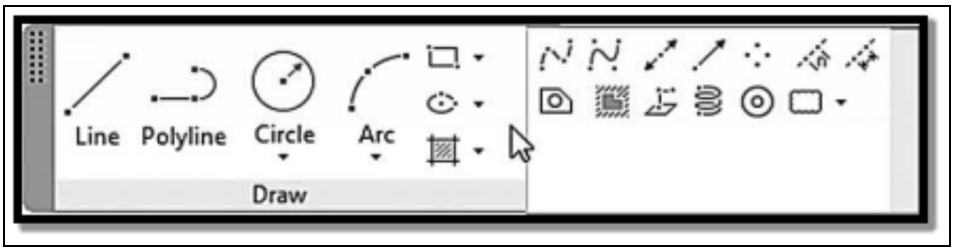
There are two states for panels in AutoCAD— either docked or floating. By default, all panels are docked in their respective tab. Move the panel in the graphical area to detach it and make it float. An essential aspect of enabling the panel to float is the ability to view it while other tabs are active. To send the panel back to its respective tab, simply click the **small button** located at the top right side.



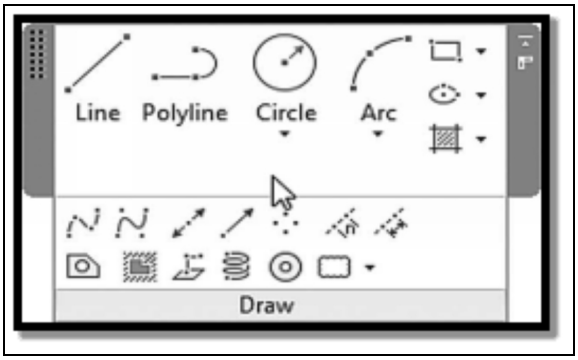

When the panel is floating, you have the option to toggle the orientation.



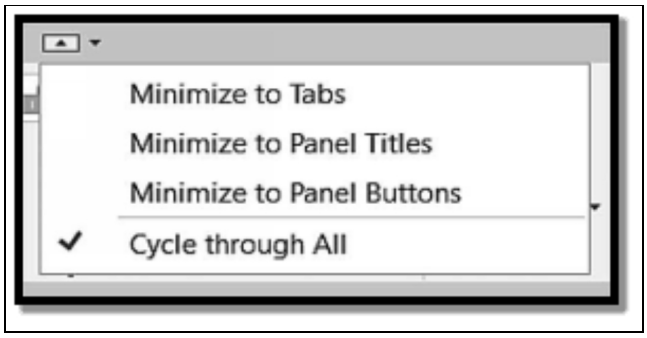

The extension will be towards the right.




Alternatively, you can move in a downward direction:

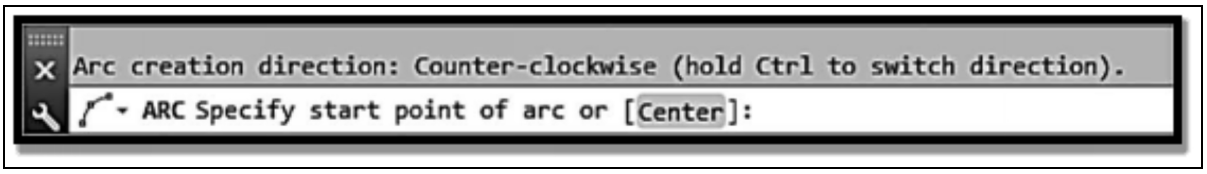


To navigate through the various states of the ribbons, simply use the button located at the end of the tab names. This new feature aims to provide users with an expanded graphical area. By selecting the small button, you will be able to generate the following outcome:




# Command Window

The command window in AutoCAD can be easily docked at the bottom or top of the screen, although it is initially set to float. Constantly monitoring the command window can greatly optimize workflow by eliminating the need to decipher AutoCAD's instructions, thus saving valuable time. In the command window of AutoCAD, you will be able to see two things: the commands you enter and the prompts from AutoCAD that guide you to perform specific actions, such as specifying a point or inputting an angle. Please consult the accompanying diagram:




# InfoCenter

The InfoCenter is the ultimate resource for accessing help topics both online and offline, as well as a variety of other essential tools.

When you enter a word or phrase in the designated field, AutoCAD will open the Autodesk Exchange window and locate all the relevant online and

offline topics for you. Use the Sign-in option to gain access to your Autodesk Account.

# Graphical Area

The graphical area serves as your drafting space. This is the designated space for creating precise and accurate drawings of lines, arcs, and circles. AutoCAD provides a highly accurate and precise environment, featuring a three-dimensional XYZ space for 3D designs and an XY plane for 2D drawings. Coordinates can be easily monitored in the left section of the status bar.

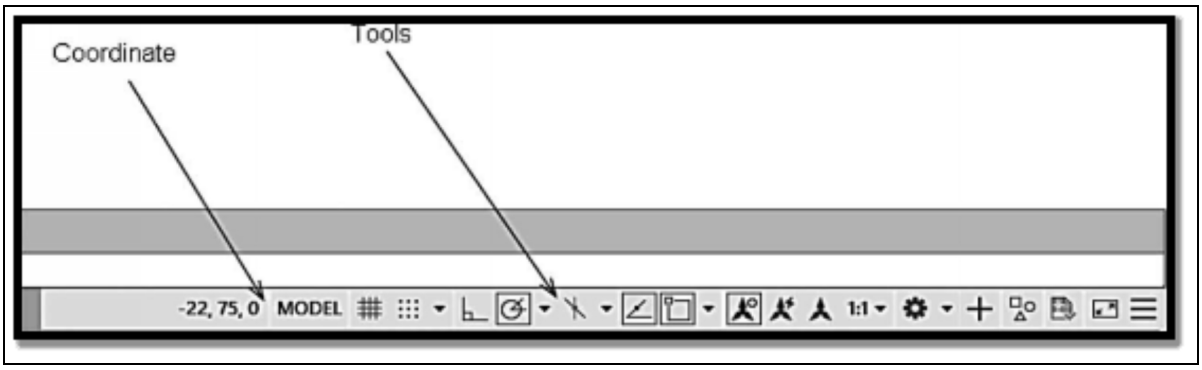
# Start Tab

Clicking the **Start tab** will bring up the welcome screen whenever you need it. Therefore, you have the option to begin a new drawing, access a pre-existing drawing, and more. On the left side of the Start tab, you will notice three horizontal lines. **By clicking on them, a menu will appear displaying the following options:**

This menu allows you to perform the same file functions as the main menu.

# Status Bar

The status bar in AutoCAD displays coordinates and various essential functions, catering to both precise 2D drafting and 3D design.



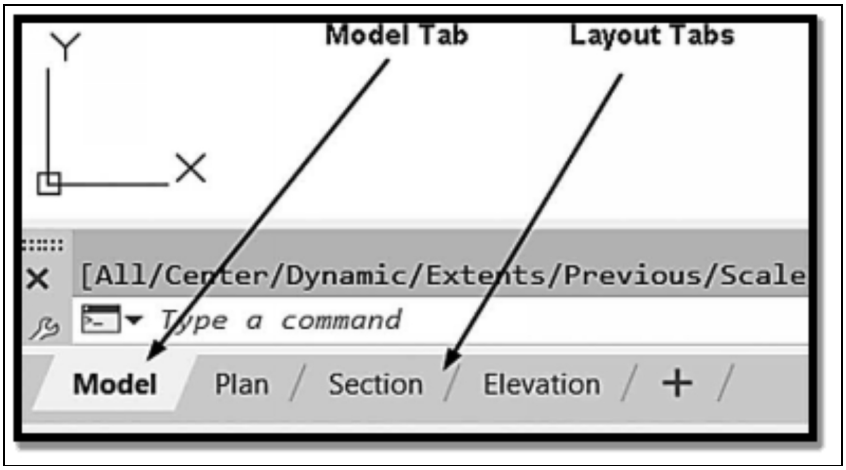

There may be additional buttons on the status bar that you haven't noticed yet. To personalize this, locate the final button on the right side (identified by three horizontal lines) and access the subsequent menu (displaying a selection of available choices)

# What Is My Drawing Unit?

When you draw a line that is 6 units long in AutoCAD, the term "**6 units**" refers to a specific measurement within the software. AutoCAD uses a unit of measurement that you have defined in your drawing settings. It could be inches, millimeters, feet, or any other unit of measurement. So, when AutoCAD says "**6 units**," it means that the line you have drawn is 6 times the length of the unit you have set in your drawing settings. What are the units of measurement: 6 meters, 6 feet, or something else? AutoCAD is capable of working with any units of measurement you prefer. If you want to represent a distance of 6 meters, you can easily do so without any limitations. Assuming you are referring to a measurement of 6 ft, AutoCAD will also make this assumption. Consistency is crucial when working on your file. When working in the Model Space, it is important to keep in mind

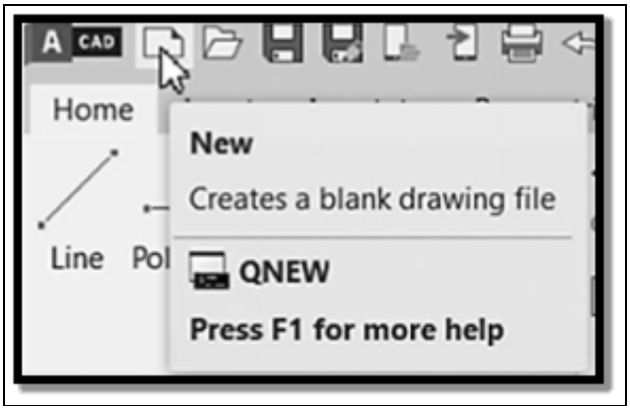
that your drafting will need to be adjusted for printing. You will need to reconsider your assumptions and set the drawing scale accordingly.

**Displayed on the lower left corner of the screen are the model tab and the layouts, as illustrated below:**

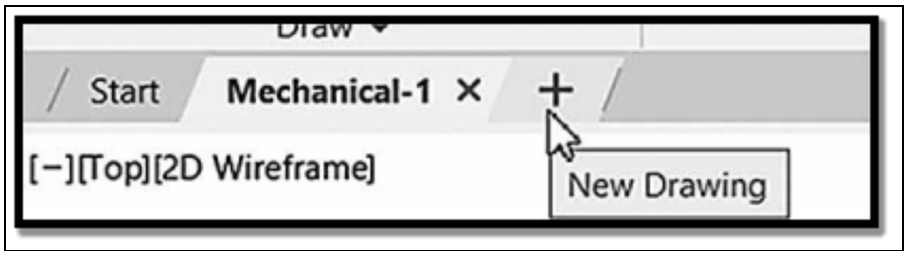



# How to Create a New AutoCAD Drawing

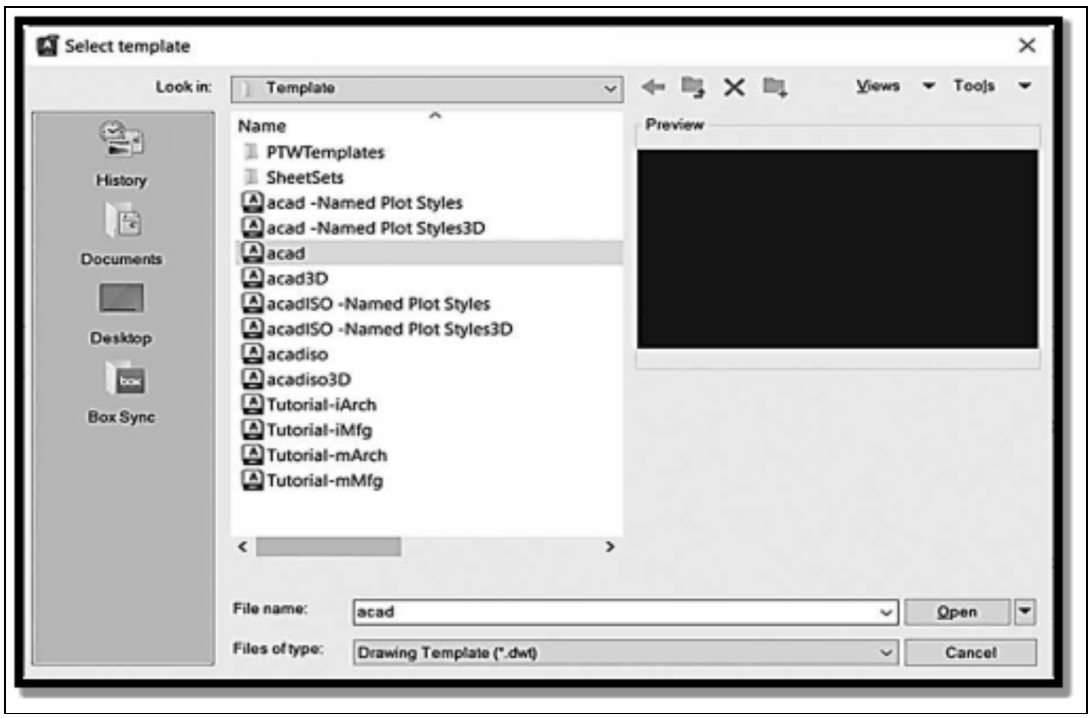
Using this command, you can effortlessly generate a fresh drawing by utilizing a preexisting template. Use the Quick Access Toolbar and select the **New button**:




Alternatively, you can access the desired option by clicking on the **(+) sign** located in the File tab.




**A dialog box will be displayed for your reference:**




**Follow these steps:**

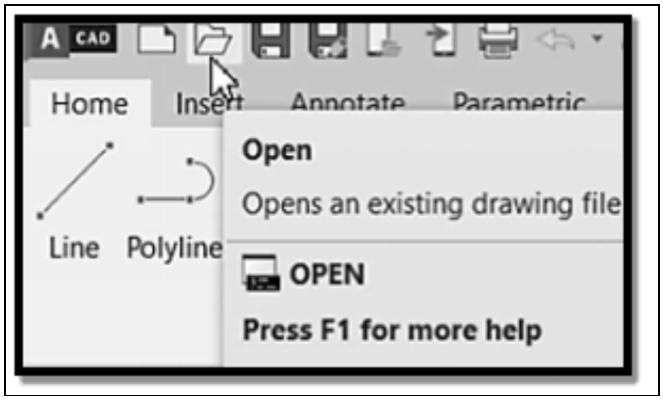
- Choose the template file you want to use. AutoCAD template files typically have the extension *.dwt. AutoCAD 2025 offers a wide range of pre-designed templates for your convenience. However, it is recommended for companies to create their template files to suit their specific needs.
- After completing your task, simply click on the **Open button**.
- AutoCAD drawing files are typically saved with the file extension *.dwg.
- When starting AutoCAD, a new file will be created with a temporary name like Drawing1.dwg. It is recommended to rename the file to something more meaningful.

**Note**

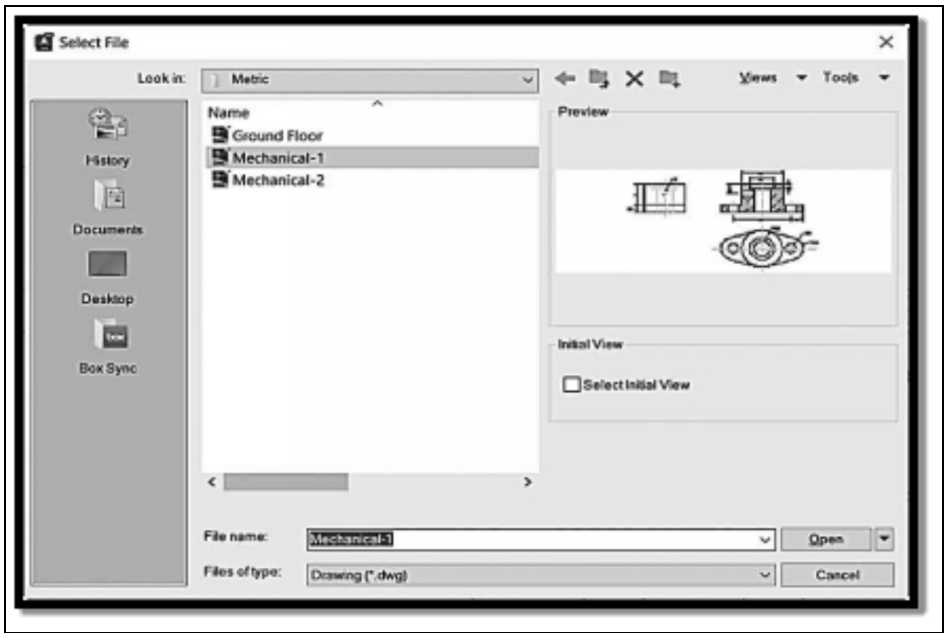
**On the Start tab, you can achieve the same result by selecting the New drop-down menu, as demonstrated below:**

# How to Open an Existing AutoCAD Drawing

To open an existing drawing file for further modifications, use this command. To open a file, simply click on the **Open button** located in the Quick Access Toolbar.




**A dialog box will be displayed for your reference:**




**Follow these steps:**

- Specify the drive and folder you want to use.
- To open a single file, simply select its name from the list and click the Open button. Alternatively, you can double-click on the file's name. To open multiple files in AutoCAD, simply select the first file name, hold down the **[Ctrl] key**, click on the other file names in the list (a common skill in MS Windows), and then click the **Open button**.

# How to Save Your Drawings

Regularly saving your work is essential in any software, and AutoCAD 2025 is certainly no exception. AutoCAD offers a wide range of powerful tools and features that enable users to effortlessly create intricate drawings that demand precision and dedication. Having a comprehensive understanding of the different techniques and optimal strategies for saving your drawings is crucial to avoid any potential data loss and guarantee the accurate preservation of your work.

# Essential Techniques for Saving in AutoCAD

## Using the Save command

- Locate and click on the "**Application Menu**" icon, which is represented by a red "**A**" in the top-left corner of the screen.
- Choose the "**Save**" option from the drop-down menu.
- When saving the drawing for the first time, AutoCAD will ask you to provide a file name and choose a location. Select an appropriate name and location for your file.
- Select the "**Save**" option to finalize the process.

## Using the Save As command

- First, locate and click on the "**Application Menu**".
- Choose "**Save As**" from the drop-down menu.
- When you reach the "**Save Drawing As**" dialog box, you have the option to choose a new file name, location, or file format (such as DWG, DWT, or DXF).
- To create a new file with the specified parameters while keeping the original file intact, simply click on the "**Save**" button.

## Mastering Keyboard Shortcuts

- To quickly save the current drawing, simply press **Ctrl + S**.
- To open the **"Save As"** dialog box, simply press **Ctrl + Shift + S**.

## Save Options and Settings

AutoCAD 2025 offers a wide range of options and settings for customizing the way your files are saved. These features can optimize workflow efficiency and guarantee seamless compatibility across various AutoCAD versions.

# AutoSave

**AutoSave is a valuable feature that ensures your work is automatically saved at regular intervals.**

- To enable or adjust AutoSave, access the command line by typing **OPTIONS** and pressing **Enter**.
- Navigate to the "**Options**" dialog box and select the "**Open and Save**" tab.
- To enable AutoSave and set the time interval, go to the "**Automatic Save**" section. From there, you can choose how often you want AutoCAD to save your work, such as every 10 minutes.
- Apply the changes by clicking "**OK**".

# Compatibility with different file formats

**AutoCAD 2025 offers the option to save drawings in various DWG file formats, ensuring seamless compatibility with previous versions of AutoCAD.**

- Open the "**Save Drawing As**" dialog box and locate the drop-down menu labeled "**Files of type**."
- Choose the preferred DWG format, such as AutoCAD 2013 or AutoCAD 2010.
- Save the drawing in the chosen format by clicking on the "**Save**" button.

# Backup files

- AutoCAD creates backup files (.bak) automatically to ensure data is not lost.
- Backup files are typically stored in the same directory as the original drawing.
- To restore a backup, you can easily change the file extension from .bak to .dwg and then open it in AutoCAD.

# Cloud Integration and Collaboration

AutoCAD 2025 provides advanced cloud integration capabilities, allowing for effortless collaboration and convenient access to your drawings from any location.

# Saving to Autodesk Docs

**Autodesk Docs is a cloud-based document management platform seamlessly integrated with AutoCAD.**

- Access the Application Menu and choose the "**Save As**" option.
- Open the "**Save Drawing As**" dialog box and choose "**Autodesk Docs**" from the "**Save in**" drop-down menu.
- Select the project folder you want and click on the "**Save**" option.

# Using the AutoCAD Web and Mobile Apps

**AutoCAD 2025 now allows users to conveniently save and access their drawings using the AutoCAD web and mobile apps.**

- Make sure your drawing is saved to a cloud storage service such as Autodesk Docs, Google Drive, or OneDrive.
- Access the AutoCAD web app (web.autocad.com) or mobile app and log in using your Autodesk account credentials.
- Easily access and make changes to your drawings directly from your cloud storage.

# Sharing Drawings

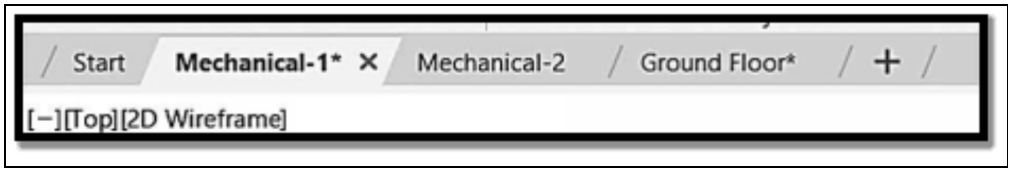
**AutoCAD 2025 now offers a convenient "Share" feature that allows for seamless collaboration.**

- Locate the "**Share**" button located in the upper-right corner of the AutoCAD window.
- Select the option to share a link or invite collaborators through email.

- Configure the desired permissions (view or edit) and then select the "**Share**" option.

# Using the File Tab

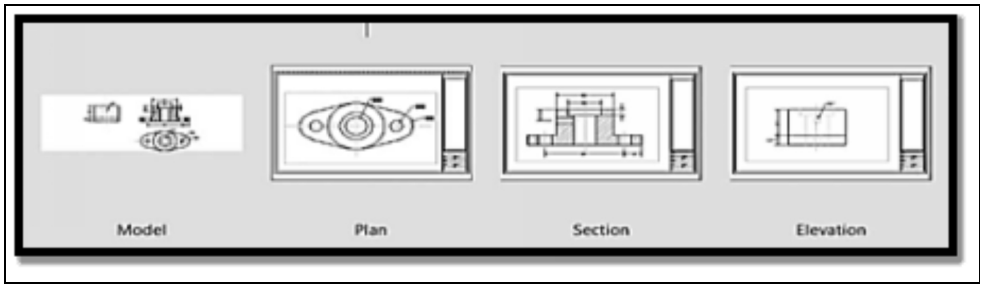
**Underneath the ribbon, you'll find the File tab which displays a tab for each opened file, like this:**




The color selected is gray. The active file (tab) will be displayed in a lighter shade of gray, while the remaining tabs will be displayed in a darker shade of gray. **When you hover over one of the tabs, two things will happen:**

- The file path will be displayed.
- The model space and the layouts of this file will be displayed below.

**Take a look at the following:**




The model space view will have a blue frame surrounding it. To access the layouts in the graphical area, simply move your mouse to the right. Once you locate the desired layout, just click on it to navigate to that view.

# How to Close Drawing Files

This command allows you to close either the currently opened files or all opened files, depending on your selection. To close a file in AutoCAD, navigate to the **Application Menu** and hover your mouse over the **Close**

**button**. From there, you can choose to close the current drawing or close all opened files with a single command.

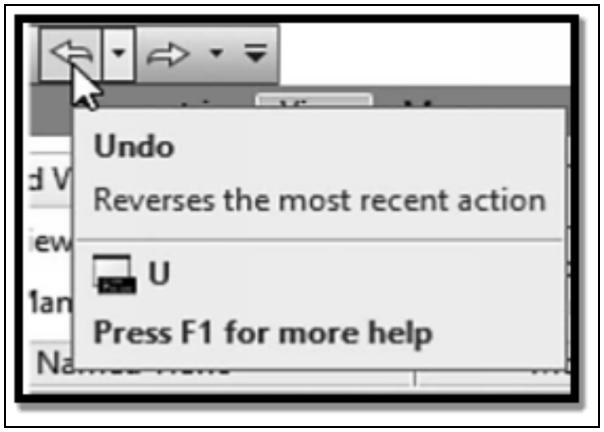
When any of the open files have been modified, AutoCAD will prompt you with a dialog box, giving you the option to save the changes or close without saving.

# Using the Undo and Redo commands

Undo is a powerful command in AutoCAD that allows you to reverse your previous actions. It is an essential tool for any AutoCAD expert. With the Undo command, you can easily correct any mistakes or make changes to your drawing with confidence. Simply type "**Undo**" or use the keyboard shortcut to access this feature

# Undo command

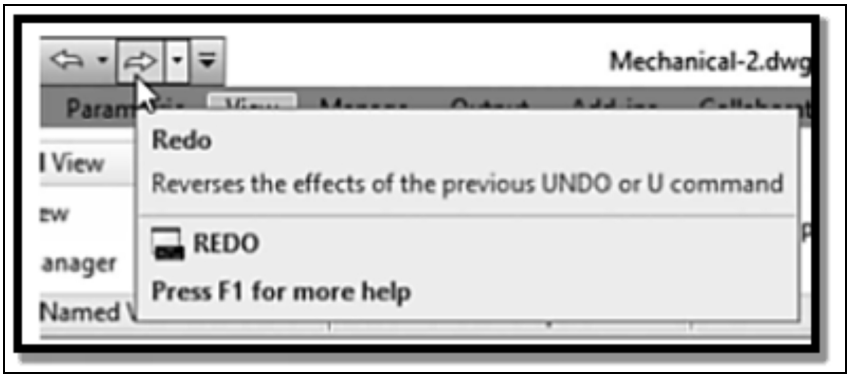
This command will reverse the effects of the previous command. To access this command, navigate to the Quick Access toolbar and select the **Undo button**. To undo multiple commands, locate and click the small arrow on the right. **First, you'll find a comprehensive list of commands. From there, simply choose the desired group and proceed to undo them:**




In addition, you have the option to enter 'u' in the command window (avoid typing '**undo**' as it has a distinct purpose) or use the keyboard shortcut **Ctrl + Z**.

# Redo command

This function will reverse the previous action. To access this command, navigate to the Quick Access toolbar and locate the Redo button. To redo multiple commands, simply click on the small arrow located on the right. Once you access the command list, simply choose the desired group and proceed to execute the commands again.



In addition, you have the option to enter the "**redo**" command in the command window or use the shortcut **Ctrl + Y**.

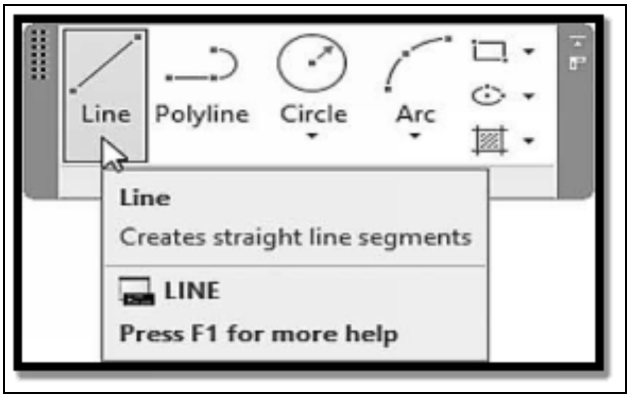
# CHAPTER TWO
# AUTOCAD DRAFTING

## What are the Drafting Priorities

When drafting, two primary considerations come to mind: ensuring the precision of your drawing and maximizing your drafting speed. Efficiency and precision are top priorities when it comes to completing drawings. Professionals in the field of AutoCAD prioritize precision over speed when it comes to their work. Accuracy is crucial when considering the entire lifespan of the drawing. If your drawing is precise, it will enable future modifications to be carried out smoothly by other individuals. However, if you complete your drawing quickly but the objects are not accurate, it will greatly negatively impact their lives.

## How to Draw Lines Using the Line Command

This command allows you to effortlessly create straight lines, with each line segment representing a distinct object. To execute this command, navigate to the **Home tab**, find the **Draw panel,** and click on the **Line button**:




**These are the prompts that will be displayed in AutoCAD:**

Specify the first point:

Specify next point or [eXit/Undo]:

Specify next point or [Close/eXit/Undo]:

Specify next point or [Close/eXit/Undo]:

**For the initial prompt, please provide the coordinates of your first point. Continue specifying points until you have finished, and then choose one of the following options:**

- To stop the command without closing the shape, you can use the **X key** or press [**Enter**]. It's also possible to use [**Esc**], but it's not recommended to make it a habit since [**Esc**] typically means abort.
- To close the shape and complete the command, you can either press the **C key** on your keyboard or right-click and choose the **Close option**.
- To rectify any errors, simply press the **U key** on your keyboard or right-click and choose the Undo option.

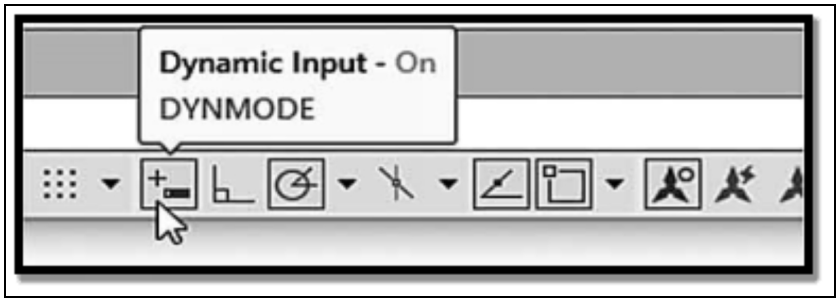
**Here is a visual representation of the right-click menu:**

# Understanding Dynamic Input in AutoCAD 2025

**Dynamic Input offers a range of functions:**

- Prompts will be displayed in the command window within the graphical area.
- Before drafting, AutoCAD will display precise measurements of line lengths and angles, ensuring accurate specifications.

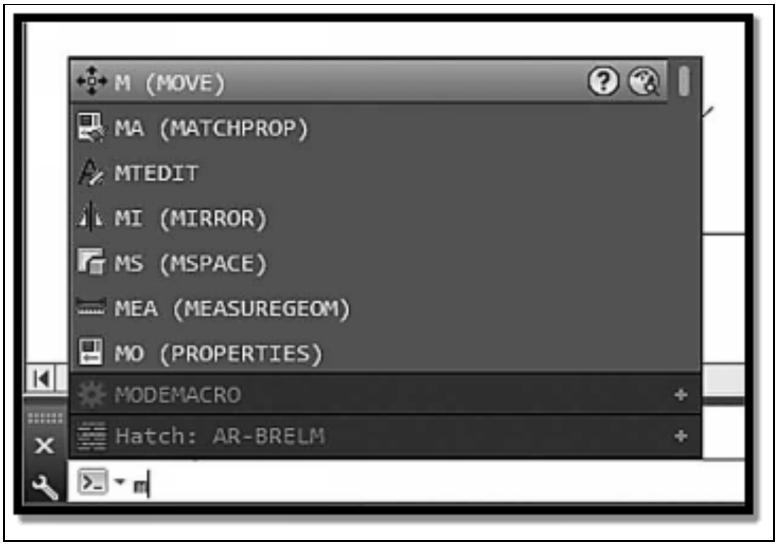
**To enable or disable Dynamic Input, simply click the corresponding button located on the Status bar.**



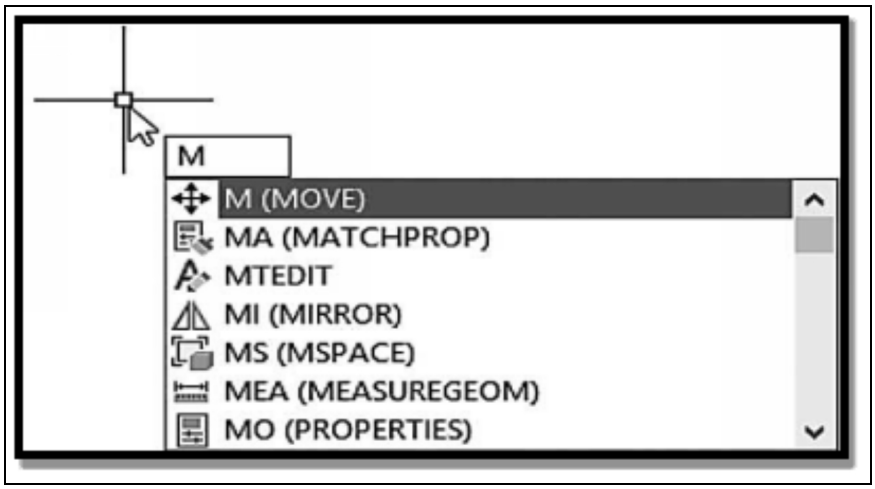

# Exercises to Enhance Line Drawing Skills with Dynamic Input

When you enter a command in the command window, AutoCAD will assist you by displaying all the commands that begin with the same letters.

**For instance, consider the following example:**

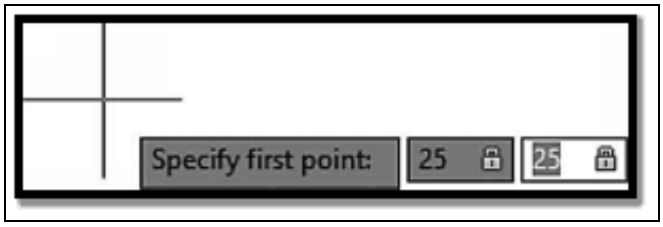



After entering the letter "**m**" in AutoCAD, the software promptly displayed a comprehensive list of commands that begin with that letter. When Dynamic Input is enabled, it also applies to the crosshairs. Here's an example:



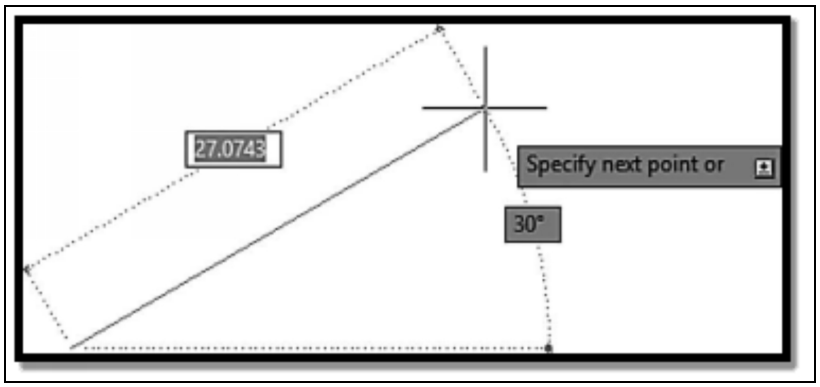

Assuming we have successfully opened the command window, using either method. After pressing [**Enter**], a prompt will appear.

Enter the x and y coordinates, using the [**Tab**] key to navigate between the two fields:




# Illustration of Specifying Lengths and Angles

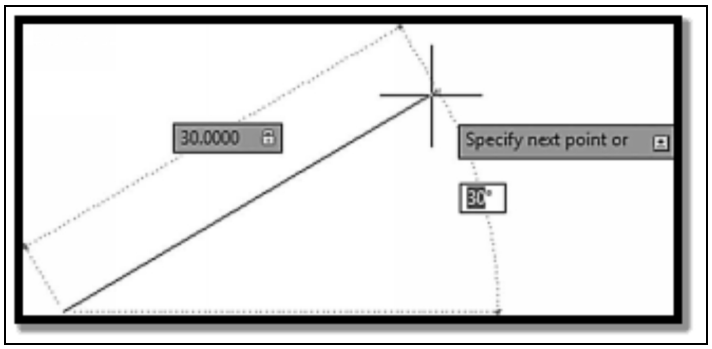
After specifying the starting point, AutoCAD will use Dynamic Input to display the length and angle of the line using rubber band mode.




**Take note of the following:**

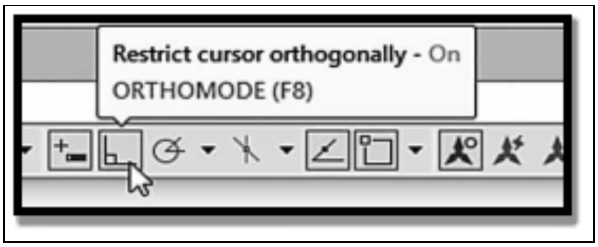
- Angles in AutoCAD are measured in a full 360° rotation, unlike the system where angles are measured counterclockwise starting from the east for only 180°.

- Enter the length of the line and use the [**Tab**] key to input the angle in 1° increments. Press [**Enter**] when you are finished to specify the first line. Proceed with the identical process for the remaining segments:

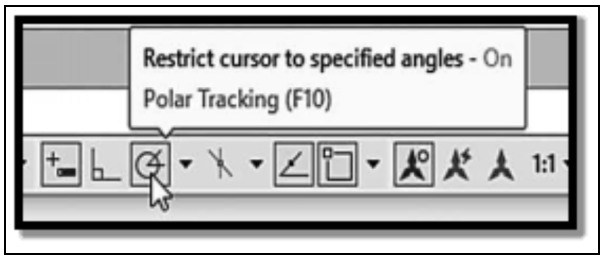



# Exact Angles (Ortho vs. Polar Tracking)

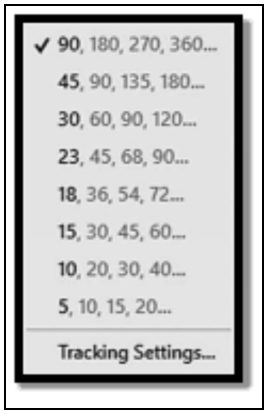
When using Dynamic Input, angles are incremented by 1°. However, it is important to note that relying solely on this feature may not guarantee precise angle specifications in AutoCAD. The Ortho function ensures that the lines are perfectly perpendicular by restricting them to angles of 0, 90, 180, and 270 degrees. To enable or disable Ortho in AutoCAD, simply click on the corresponding button located in the status bar.




But what if we need to use different angles like 30, 45, 60, and so on? In this scenario, using Ortho won't be beneficial. However, AutoCAD has a solution called Polar Tracking. This feature displays rays in the graphical area, originating from the current point and aligned with specific angles such as 30, 45, and so on. Enabling Ortho or Polar Tracking will automatically disable the other, as they are mutually exclusive. To enable or disable Polar Tracking, simply locate the corresponding button on the status bar.




**By clicking the small triangle located at the right of the button, a menu will be displayed:**




To modify the default settings of Polar Tracking, you have the option to either select the desired angle or access Tracking Settings.

# Increment Angle

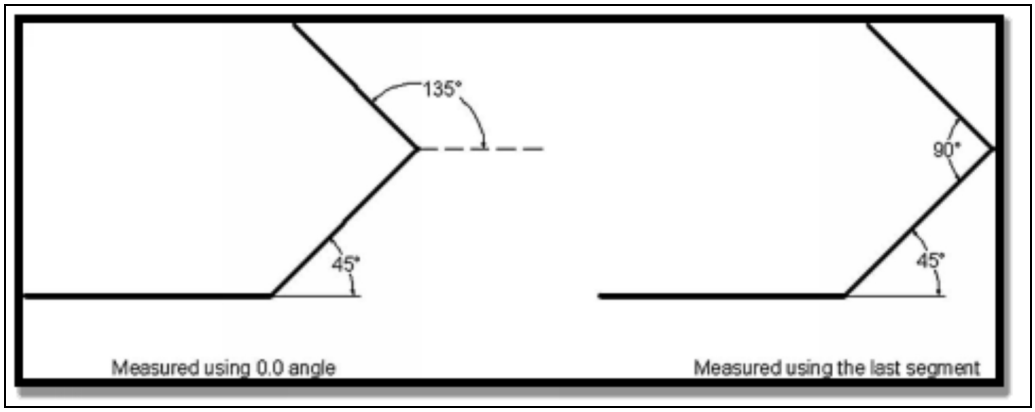
The increment angle is the angle that should be used, along with its multiples. Choose an option from the provided list or enter your own.

# Additional Angles

If you set your increment angle to 30, Polar Tracking will not allow you to use 45 as one of the angles. Therefore, it is necessary to designate it as an extra angle. It is important to note that you will not use its multiples.

## Polar Angle Measurement

When using Polar Tracking in AutoCAD, you have the option to define angles as either absolute angles (with 0° at the east) or use the previous line segment as your reference for the 0 angle. Check the accompanying diagram:




When using either Ortho or Polar Tracking, you can manipulate the angle. Simply input the desired distance and press [**Enter**] to ensure precise measurements. This technique is commonly referred to as Direct Distance Entry.

# Precise Drafting using Object Snap

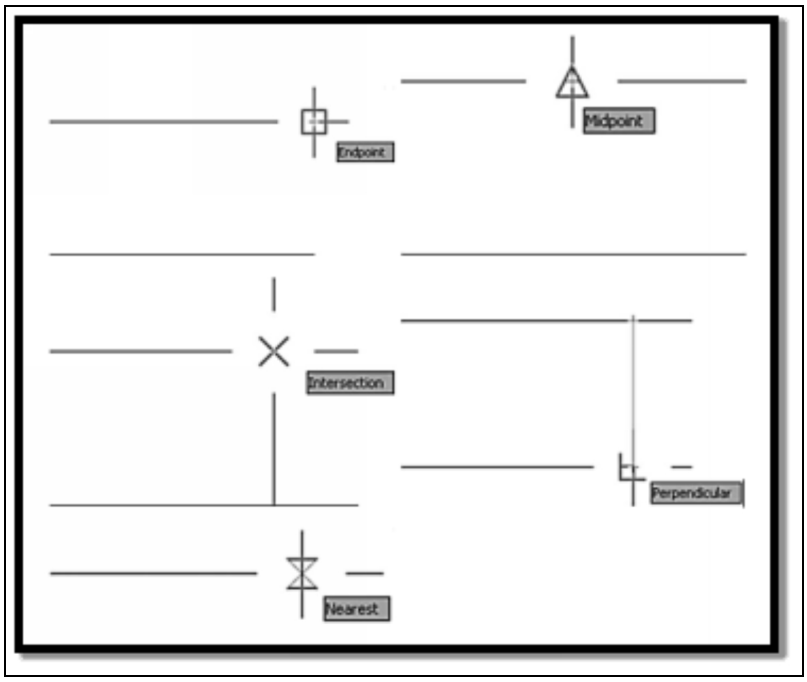
Object Snap, also known as OSNAP, is a crucial tool for achieving precise accuracy in both 2D and 3D designs within AutoCAD. AutoCAD provides a precise method for specifying points on objects by using its database stored within the drawing file.

**Here are a few examples of Object Snaps:**

- **Endpoint**: To capture the endpoint of a line or arc, you can use the Endpoint command.
- **Midpoint**: To locate the midpoint of a line or arc, use the midpoint tool.
- **Intersection**: To capture the intersection of two objects (regardless of their type)
- **Perpendicular**: To capture the perpendicular point on an object, you can use the Perpendicular function.
- **Nearest**: When you want to select a point on an object that is closest to your click point, you can use the "**Nearest**" function.

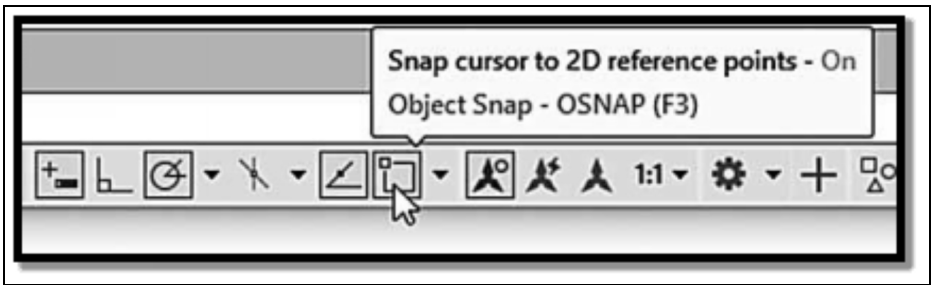
We will cover additional object snaps as we delve into different drawing objects.

**Here are visual representations of each of these OSNAPs:**

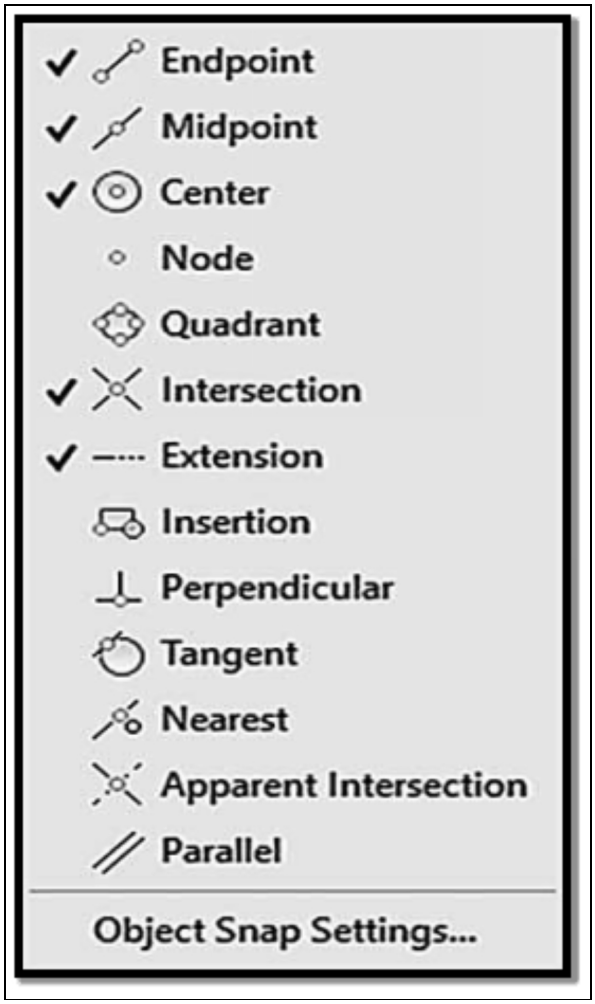



# How to Enable Running OSNAPs

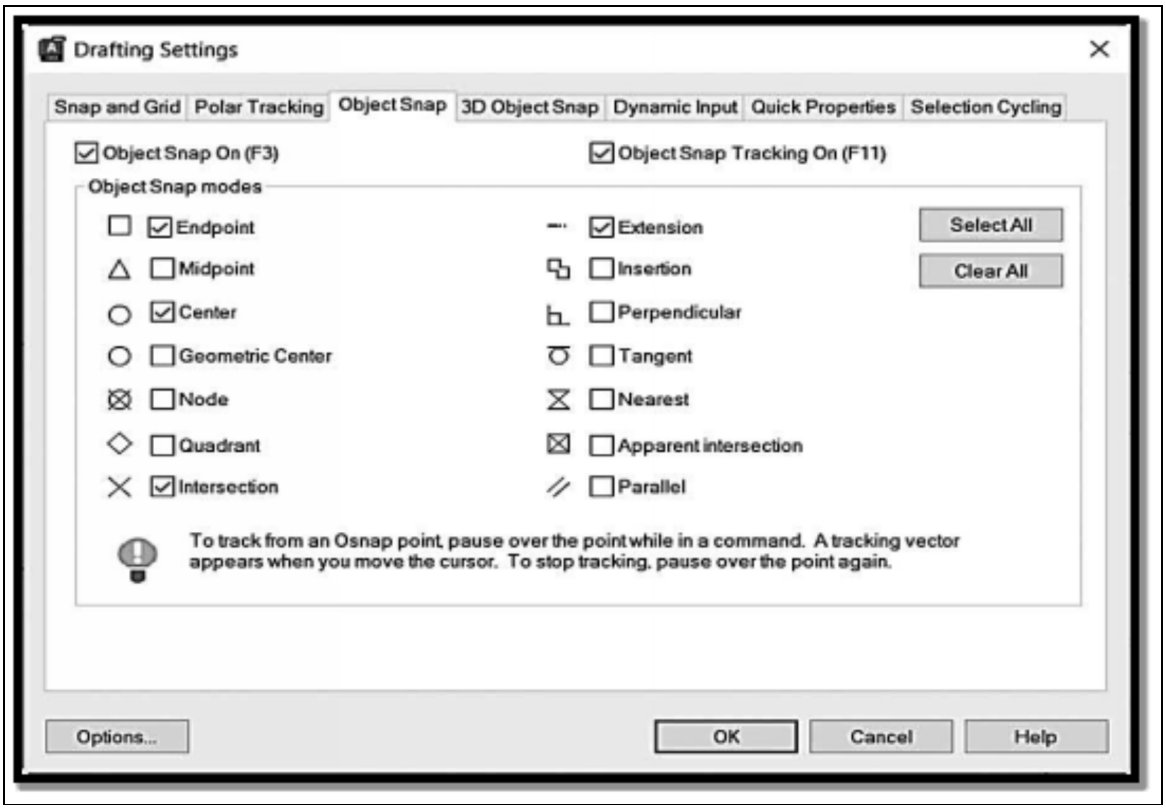
To enable running OSNAPs in the drawing, simply click on the **Object Snap button** located in the status bar.




**By clicking the small triangle located at the right of the button, a menu will be displayed:**




To enable the desired OSNAPs, you can activate them individually. To make a more convenient change, simply select the Object Snap Settings option. This will prompt a dialog box to appear.

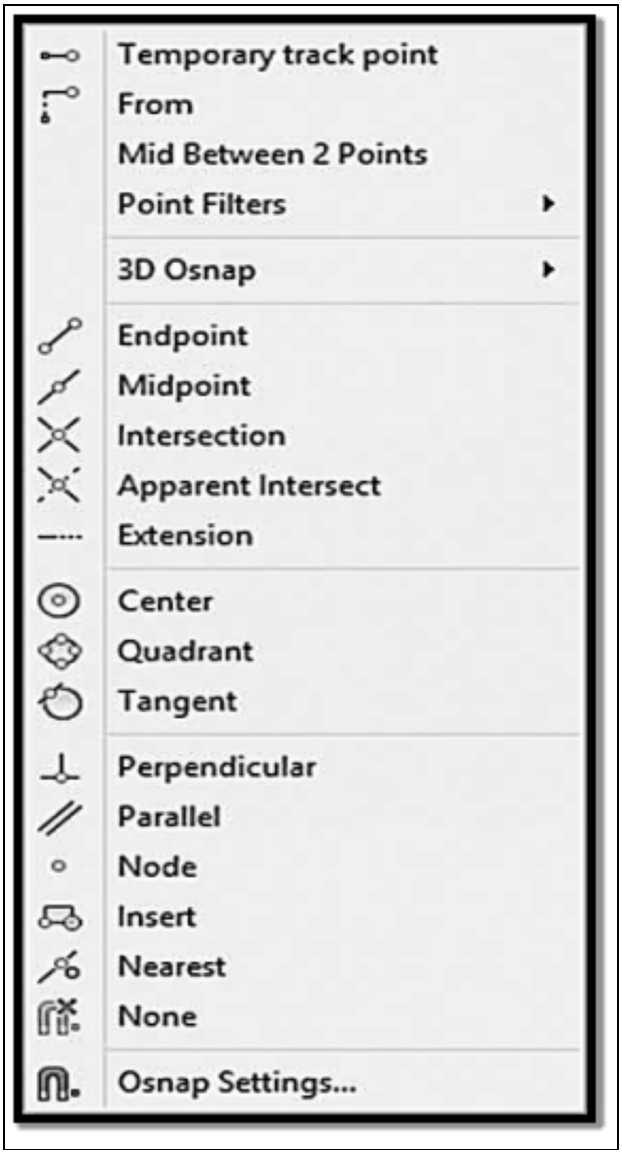



On the right side, you'll find two buttons: **Select All** and **Clear All**. It is advisable to begin by using the **Clear All command,** followed by selecting the desired OSNAPs. After completing the task, simply click on the **OK button**.

# Using OSNAP Override

When the Object Snap button is enabled, certain OSNAPs are functioning correctly, while others are not responding as expected. If you need to temporarily disable all OSNAP settings, you can do so by using a single command. Once you have completed your task, you can easily restore the settings to their original state. This feature is known as OSNAP Override.
**There are two methods to activate an override:**

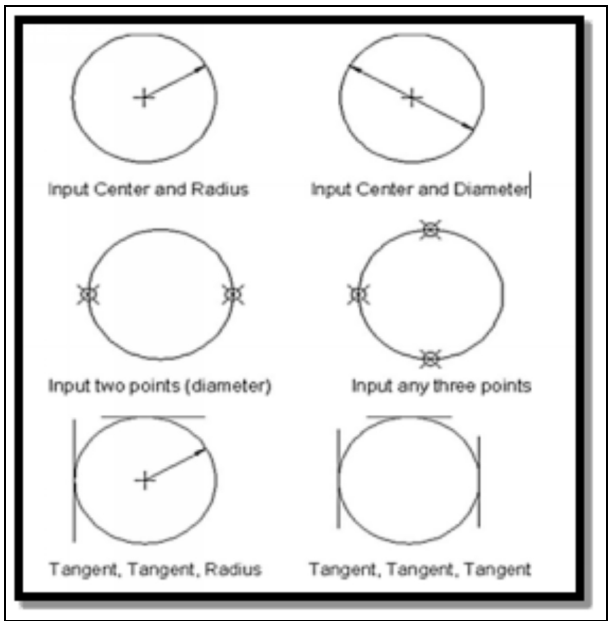
- When using AutoCAD, you can quickly access the desired OSNAP by typing the first three letters on your keyboard.
- Hold down the [**Shift**] key on your keyboard and right-click. A pop-up menu will be displayed for your reference.



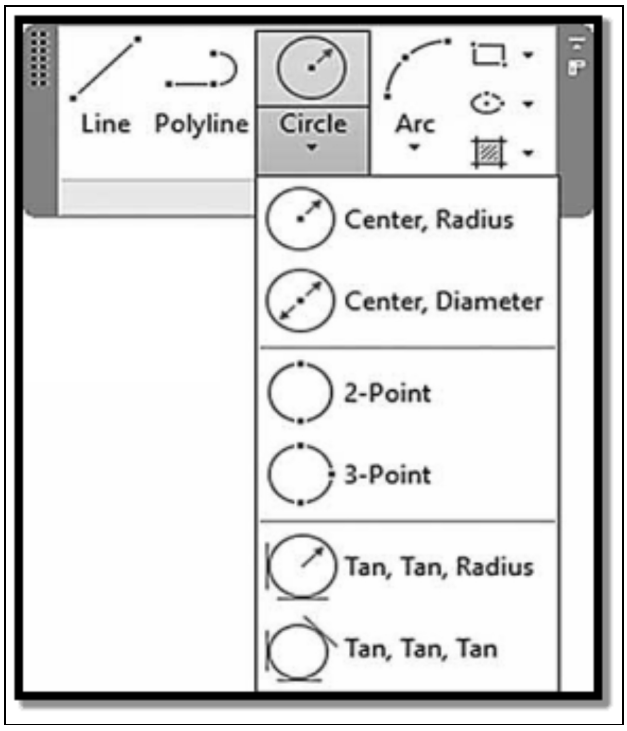

# How to Draw Circles Using Circle Command

Creating circles in AutoCAD is a breeze when using the Circle command. With this command, you can easily draw precise circles of any size and location. This command allows you to create a circle using various methods, depending on the data at hand. If you are familiar with the coordinates of the center, there are two methods you can use. If the user is familiar with the coordinates of points on the diameter of the circle, there are two additional methods available. Additionally, if there are existing drawn objects such as lines, arcs, or circles that can serve as tangents for the circles you want to create, there are two additional methods you can use.

**Here are six methods for drawing a circle in AutoCAD:**




To execute this command, navigate to the **Home tab** and find the **Draw panel**. From there, click on the dropdown arrow next to the **Circle button** to access a list of all the available **methods:**



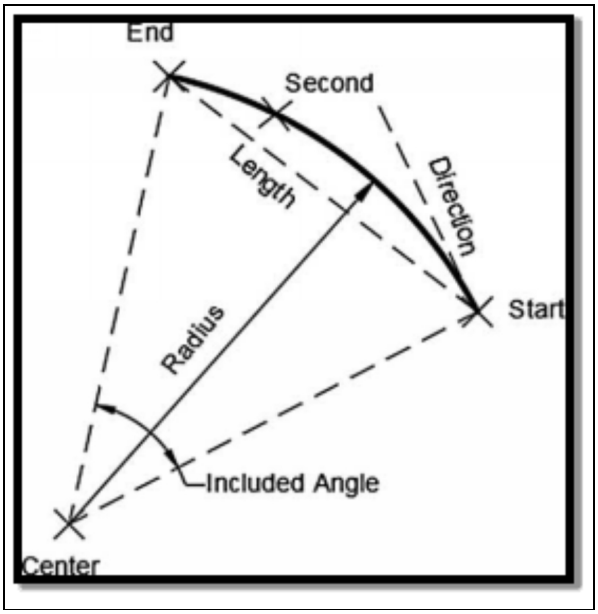
# How to Draw Circular Arcs Using Arc Command

This command allows you to create an arc, which is a portion of a circle. AutoCAD simplifies our workflow by using eight key pieces of information about a circular arc.

**Here are the following:**

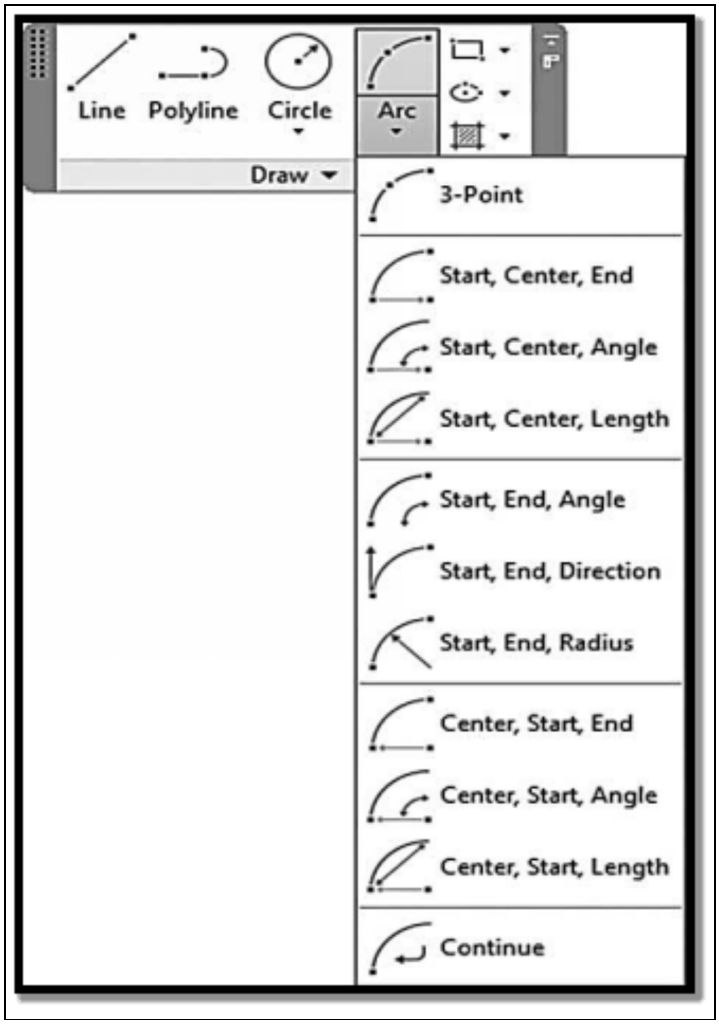
- When working with arcs in AutoCAD, it is important to determine the starting point. This will serve as the reference point for creating the arc.
- You can use any point along the arc's parameter as the second point.
- The endpoint of the arc
- The direction of the arc is determined by the tangent that passes through the start point. Enter the angle of the tangent.
- The length of the chord is the distance between the starting point and the ending point.
- The center point of the arc is a crucial element in creating accurate and precise drawings.
- The radius is an important measurement in AutoCAD.
- The angle formed by the points Start, Center, and End is referred to as the Included Angle.

**Take a look at the illustration below:**



Providing three out of the eight pieces of information will enable AutoCAD to draw an arc. However, it is important to note that not just any three pieces will suffice. To access the required information, navigate to the **Home tab** and locate the **Draw panel**.

**Then, click on the arrow next to the Arc button to reveal a list of available methods:**




It is essential to provide a starting point in every case. Typically, users are advised to consider a counterclockwise direction when specifying points. However, if you prefer to work in a clockwise direction, you can easily do so by holding down the **[Ctrl] key**.

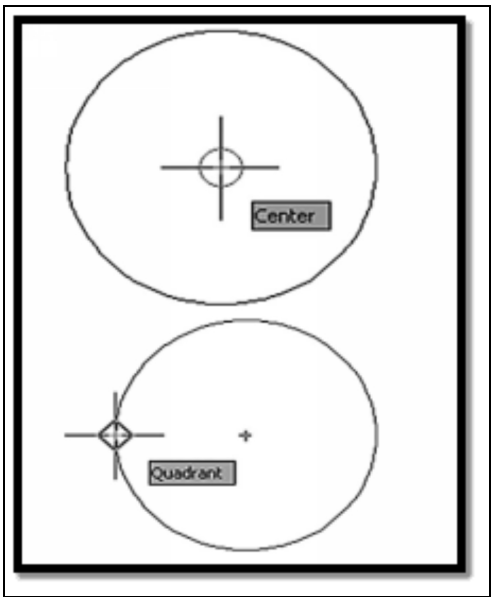
# Different Object Snaps Linked to Circles and Arcs

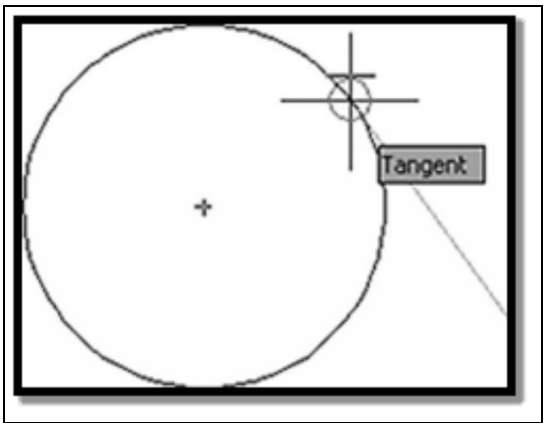
Understanding object snaps about circles and arcs is crucial in AutoCAD. These features allow for precise and efficient drawing and editing of curved elements.

**Here are a few Object Snaps that are relevant to circles and arcs:**

- **Center**: To locate the center of an arc or circle, you can use the "**Center**" command.
- **Quadrant**: To determine the quadrant of an arc or circle
- **Tangent**: To determine the tangent of an arc or circle

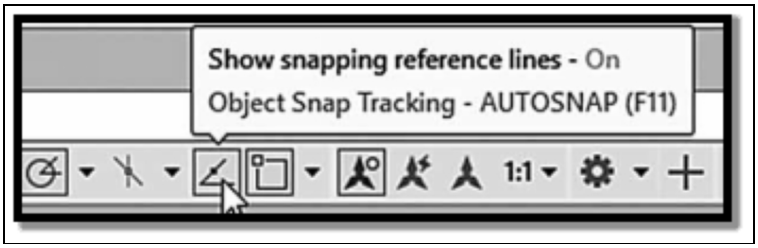
**These are the graphical representations of each OSNAP:**



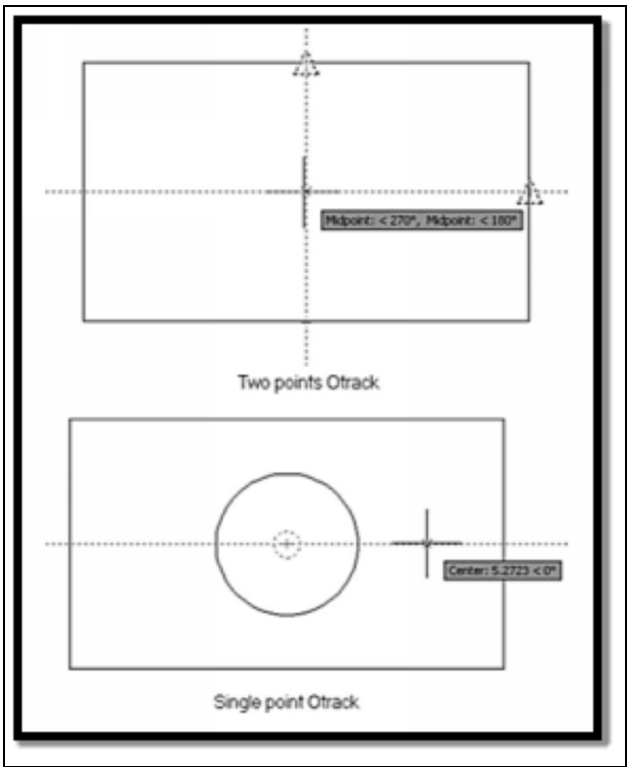
# How to Use Object Snap Tracking with OSNAP

At times, relying solely on OSNAP may not suffice when it comes to specifying intricate points. In the past, a common technique for addressing this issue was to create dummy objects as a means of accurately defining complex points. For example, when needing to specify the center of a circle at the center of a rectangle. To achieve this, a line can be drawn by selecting the midpoints of the two vertical lines and repeating the process for the horizontal lines. With the implementation of Object Snap Tracking, or OTRACK, in AutoCAD 2000, the need for drawing dummy objects has significantly decreased. To use OTRACK effectively, it is important to have the necessary OSNAP modes activated. For example, if you wish to use the midpoint, make sure to enable the midpoint OSNAP mode before using **OTRACK**. To activate **OTRACK**, you can easily access it by navigating to the status bar and locating the **Object Snap Tracking button**. Simply click on it to enable this useful feature.

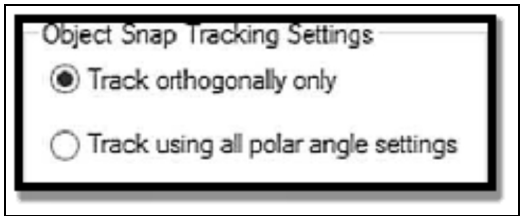



**The process is straightforward:**

- When using OSNAP, navigate to the desired point and pause for a moment without clicking. Then, move in the desired direction (either right, left, up, or down) to observe an infinite line extending in both directions. The orientation of this line will be either horizontal or vertical, depending on your movement.
- To specify a point using a single point, simply move in the desired direction, enter the desired distance, and press [**Enter**].
- When you require two points, proceed to the next point and pause briefly before moving in the desired direction. You will see another infinite line. Locate the point of intersection between the two infinite lines to determine the desired point.




**The Polar command will play a significant role in this scenario. Now, let's return to the identical dialog box:**




**Within the Object Snap Tracking Settings, there are two available options:**

- By default, the tracking in AutoCAD is set to orthogonally only.
- Use all polar tracking settings to accurately track movements.

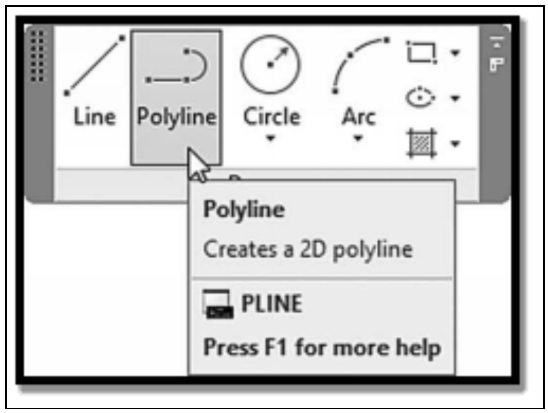
You have the option to use the existing Polar angles (increment and additional angles) to designate points using OTRACK. To deactivate an OTRACK point, simply remain at the same point for a few seconds, and it will be deactivated.

# How to Draw Lines and Ars Using Polyline Command

**The Polyline command is capable of performing a variety of tasks:**

- Create line segments and arc segments in AutoCAD.
- When creating a design, it is more efficient to use a single command to draw a complete object instead of using separate commands for lines and arcs.
- Master the art of creating lines and arcs with varying widths at their starting and ending points.

To access the command, navigate to the **Home tab** and find the **Draw panel.** From there, simply click on the **Polyline button**:




**A prompt will appear:**

- Indicate the starting point:
- The current line width is set to 1.0000.
- Provide the coordinates for the next point or choose from the available options: Arc, Halfwidth, Length, Undo, or Width.
- When using AutoCAD, you will be prompted to specify the initial point. Once you do, AutoCAD will provide you with the current line width. If you are satisfied with it, you can proceed to specify additional points using the same method we previously learned in the Line command. To modify the width, you can start by pressing the letter W or by right-clicking and choosing the

Width option. This will display a prompt with further instructions.

- Provide the desired initial width, using the appropriate format:
- Provide the desired ending width, using the format 1.0000:

First, input the initial width, followed by pressing [Enter], and then proceed to specify the final width. When you use the same file again, AutoCAD will automatically display these values for you when you use the Polyline command. To achieve Half-width, you simply need to specify half of the width instead of the full width.

- The Undo and Close options function in the same manner as those found in the Line command.
- The length parameter will determine the length of the line based on the angle of the previous segment.
- When using the Arc command in AutoCAD, a prompt will appear to draw an arc that is connected to a line segment.
- Provide the endpoint of the arc or choose from the available options such as angle, center, close, direction, halfwidth, line, radius, second point, undo, or width.

To attach the arc to the last segment of the line or make it the first object in a Polyline command, we only need two more pieces since the first point of the arc is already known. AutoCAD assumes that you can choose to reject. It assumes that the angle of the last line segment will be considered as the direction or tangent of the arc. When accepting this assumption, it is important to specify the endpoint. If not, you can select from the options below to indicate the second piece of information:

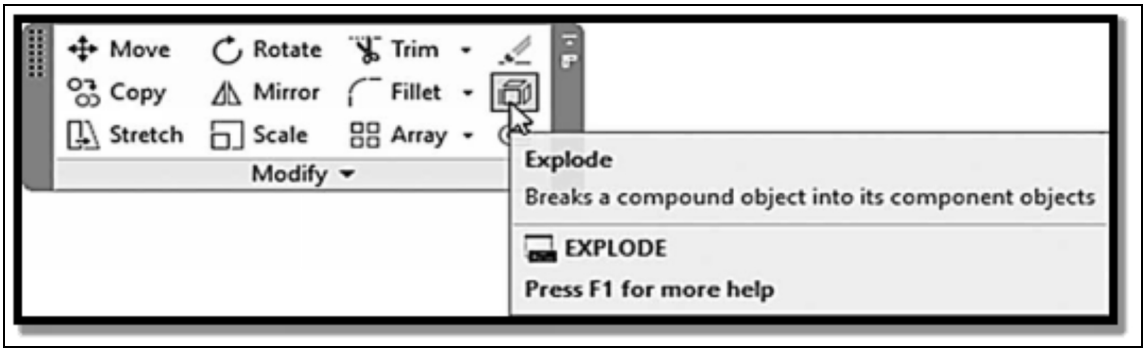
- The angle of the arc
- The arc's center point
- An alternative approach to the arc
- The arc's radius
- The second point can be any point along the arc's perimeter.

After selecting the second point, AutoCAD will prompt you to provide the third piece of information. Rest assured, it will be one of the eight options we previously discussed.

It is important to keep in mind the direction when specifying points. By default, the user should think counterclockwise. However, if you prefer to work clockwise, you can easily do so by holding down the [**Ctrl**] key.

# How to Convert Polylines to Lines and Arcs

Mastering this technique is crucial as it allows for the seamless conversion of polylines to lines and arcs, as well as the conversion of lines and arcs back to polylines. The Explode command allows you to break down a polyline into individual lines and arcs. To execute this command, navigate to the **Home tab**, find the **Modify** panel, and click on the **Explode button**:

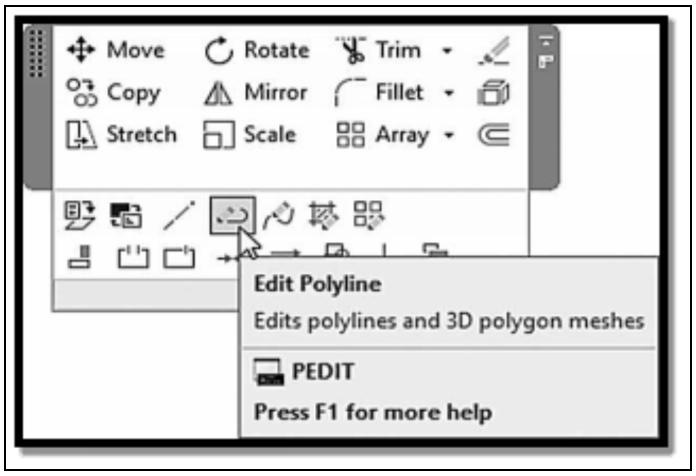



**After launching AutoCAD, a prompt will be displayed:**

- Choose the desired objects:
- Choose the polylines you want and hit [**Enter**] when you're finished. Take a look at the updated shape and you'll notice that it has been transformed into a combination of lines and arcs.

# Joining lines and arcs to form a polyline

In this discussion, we will explore the Join option found within the Edit Polyline command. To execute this command, navigate to the **Home tab**, find the **Modify panel**, and click on the **Edit Polyline button**:




**After launching the program, a series of prompts will be displayed for your attention:**

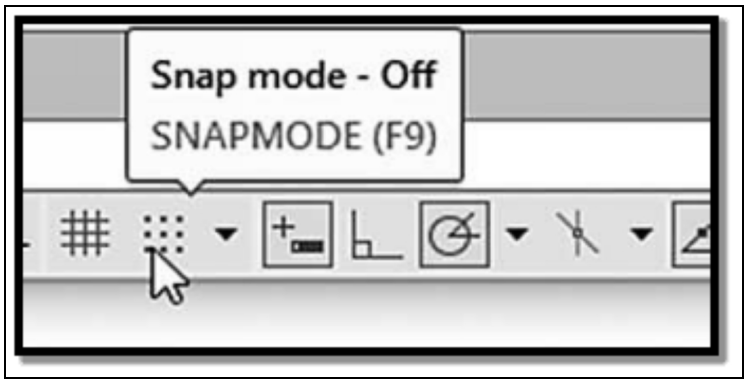
- Choose a polyline or select multiple options:
- The selected object is not a polyline.
- Select the option you want to perform from the following list: Close, Join, Width, Edit vertex, Fit, Spline, Decurve, Ltype gen, Reverse, or Undo. To choose an option, simply enter the corresponding letter.

Begin by choosing one of the lines or arcs that you wish to convert. When you select an object in AutoCAD that is not a polyline, the software will notify you and provide the option to convert the individual line or arc into a polyline. Once you accept this, a range of options will become available, including the Join option. Choose the Join option and proceed to select the remaining lines and arcs. After completing your task, remember to press the [**Enter**] key twice. After completing the task, it is advisable to review the objects. You will notice that they have been transformed into polylines.

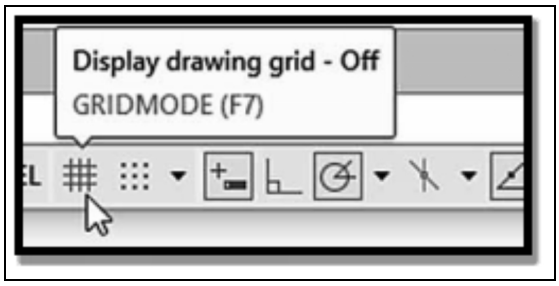
# How to Use Snap and Grid to Specify Precise Points

Using Snap and Grid is a highly effective way to ensure precise point specification in the XY plane. Typically, the mouse lacks precision, making it unreliable for pinpointing specific points. Controlling the movement is the primary purpose of Snap in AutoCAD. With Snap, you have precise control over the mouse's movement in both the X and Y planes, allowing it

to jump exact distances. The grid alone may not provide precise measurements, but it can be a useful tool when used in conjunction with the Snap function. The drawing sheets will be replicated with horizontal and vertical lines displayed. To enable or disable Snap in AutoCAD, simply click on the corresponding button located in the status bar.

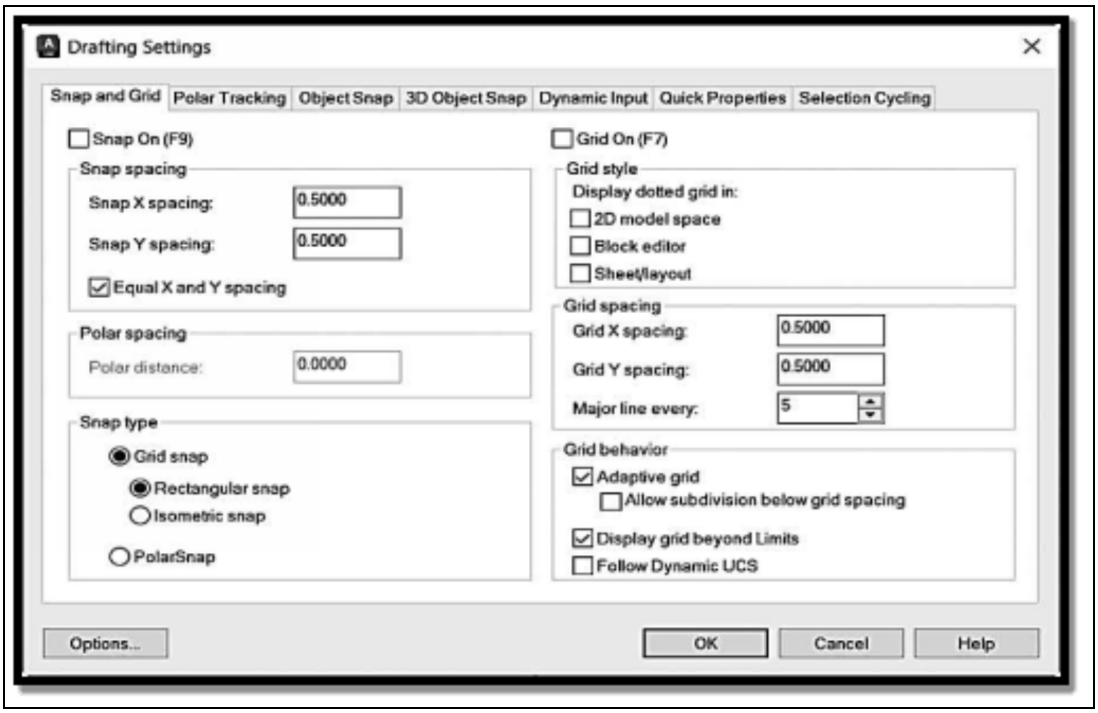



To toggle the Grid on or off, simply locate and click the corresponding button in the status bar.




To achieve the desired outcome, it is unlikely that simply turning both options on will suffice. You will need to make adjustments to the settings to tailor them to your specific needs. By accessing the small triangle located to the right of the Snap button, you will be presented with the following options:

Click on the **Snap Settings option** to open the Drafting Settings dialog box.




Provide the values for the Snap X Spacing and Snap Y Spacing. By default, these values are equal. Disable the checkbox to ensure that they are not equal. Ensure that the Grid Spacing in both the X and Y directions is set to the same value. To make the Grid align with Snap, set the Grid spacing to zero. Within the Grid, there are both major and minor lines. You can adjust the frequency of the major lines. Specify your preference for displaying the Grid as dots, resembling the previous AutoCAD 2011 version. Indicate whether you want this setting to apply to the 2D Model space, Block editor, or Sheet layout. The grid behavior is exclusive to 3D designs. Please indicate the desired Snap type: Grid Snap or Polar Snap.

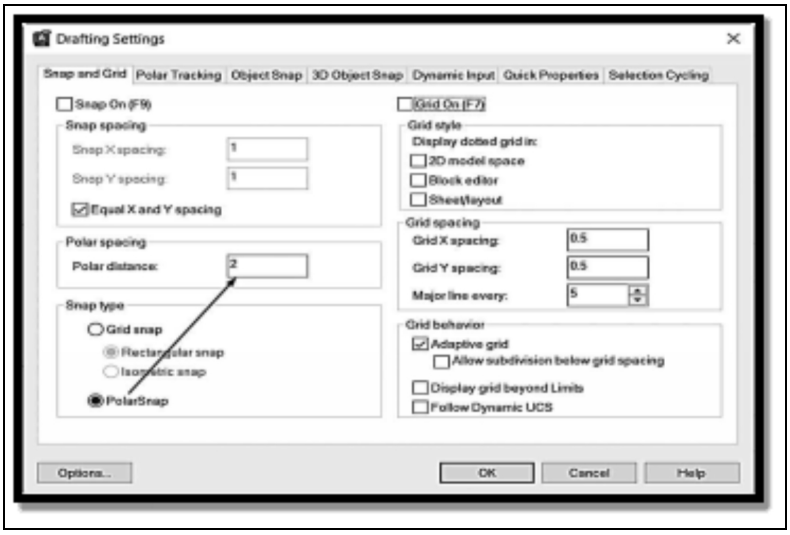
**To enable or disable both Snap and Grid, use the function keys.**

- Pressing F9 toggles the snap function on or off in AutoCAD.
- Pressing F7 will toggle the grid on or off.

# How to Use Polar Snap

When using the **Snap command,** the mouse will utilize the increment distance to precisely specify distances solely in the horizontal and vertical directions. When moving in a diagonal direction, the Snap feature won't be of assistance. AutoCAD offers a solution to this issue through the

implementation of Polar Snap. This feature allows for precise increments based on the specified angles in the Polar Tracking dialog box. Enabling the Polar Snap feature in AutoCAD will automatically disable the Grid Snap function. To accomplish this, navigate to the Snap button and click on the small triangle located on the right side of the button. From the dropdown menu, choose the **Snap Settings option**. **This action will open a dialog box displaying the following settings:**

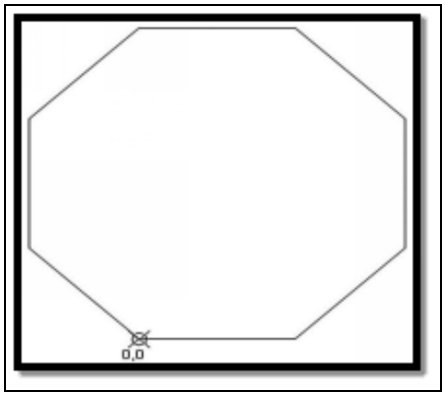



Click on the **Polar Snap button** and enter the desired Polar distance, as indicated in the illustration above.

# CHAPTER DRAWING PROJECTS

# DRAWING LINES USING DYNAMIC INPUT

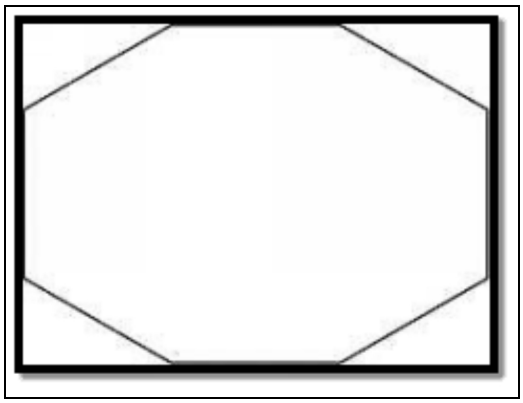
- Open the Practice 2-1.dwg file in AutoCAD 2025.
- Make sure Dynamic Input is enabled by using the status bar to click off Polar Tracking, Ortho, and Object Snap.
- With 0,0 as your starting point, draw the following shape, keeping in mind that all sides equal 4, and all angles are multiples of 45°:




- Close and save the file.

# EXACT ANGLES PRACTICE

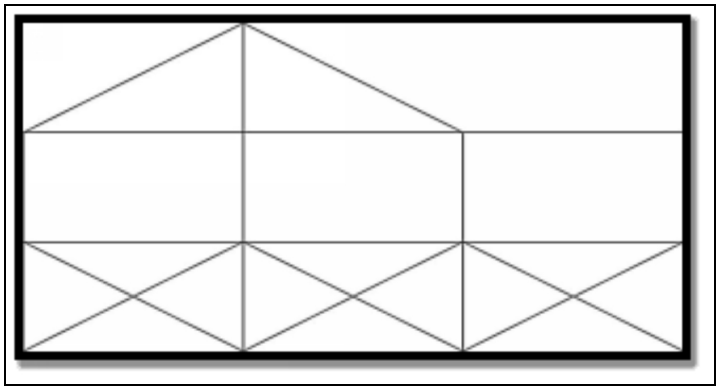
- Launch AutoCAD 2025 and select **Practice 2-2.dwg**.
- Using the Line command, draw the following form, keeping in mind that you must use **Polar Tracking** and starting at **0,0** (or any other desired point). Using the Direct Distance Entry method, enter the precise distances with all angles being 30 and their multiplications, and all lines being 2 units in length



- Close and save the file.

# SNAP (OSNAP) Objective

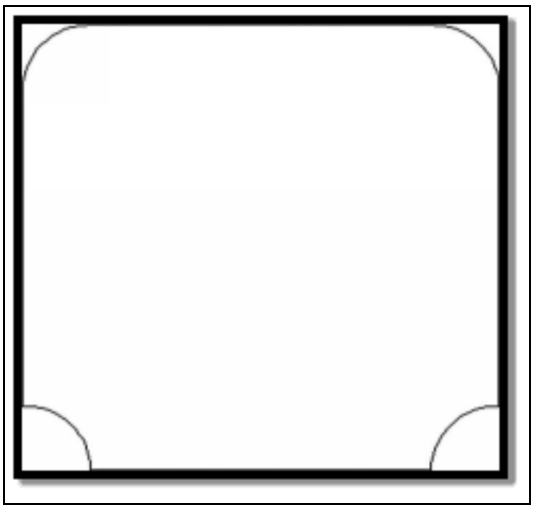
- Launch **AutoCAD 2025** and select **Practice 2-3.dwg**.
- Verify that only Endpoint, Midpoint, Intersection, and Perpendicular OSNAPs are turned on by looking at the Object Snap in the status bar.
- **Draw lines in the drawing using the Object Snap and Line command to produce the following:**



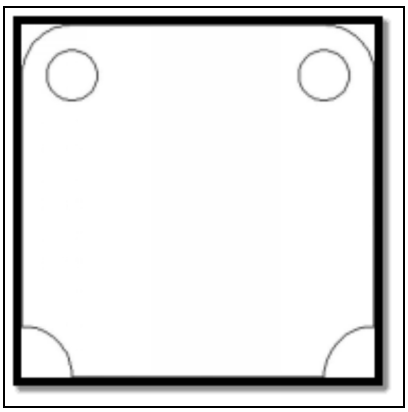
- Close and save the file.

**DRAWING USING OSNAP AND OTRACK**

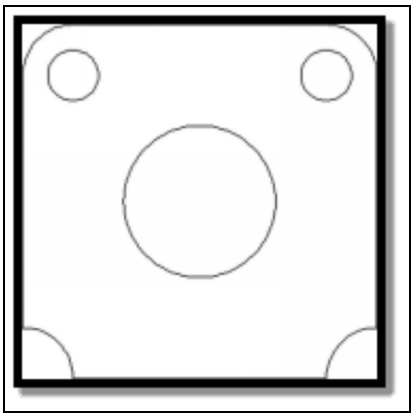
- Launch AutoCAD 2025, then select Practice 2-4a.dwg.
- Make the four arcs as indicated below, using the appropriate OSNAP and OTRACK settings:



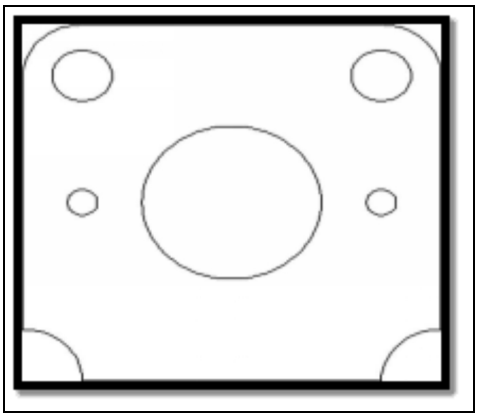
- As indicated below, draw the two circles (radius = 1).



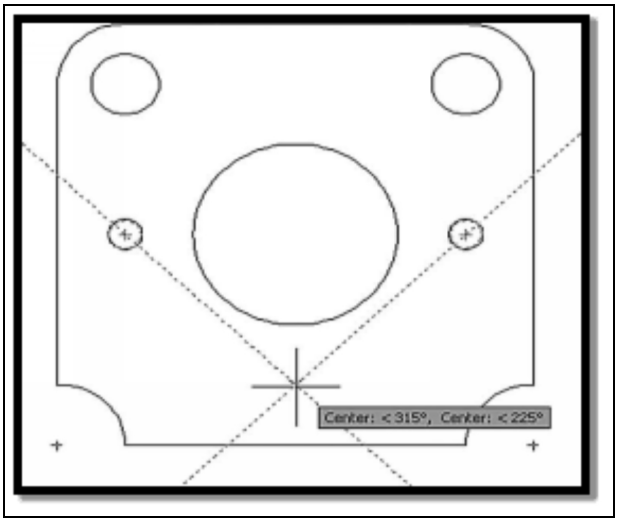
- Draw the circle in the shape's center (radius = 3.0) using OSNAP and OTRACK (using two points)<Figure>.



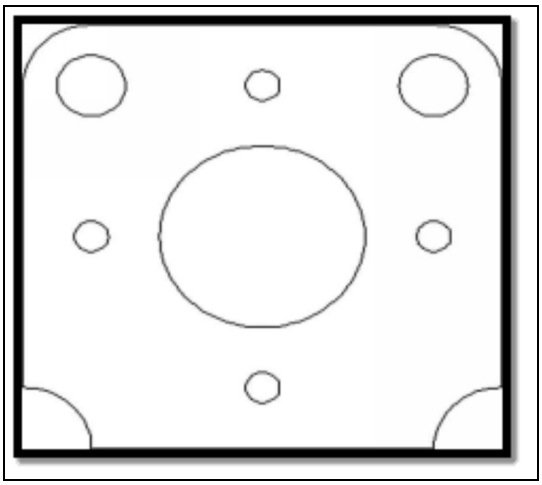
- Draw the two circles at the right and left (radius = 0.5 and center-to-center = 5.0) using OSNAP and OTRACK (one point).



- In the Polar Tracking dialog box, set the increment angle to 45. As demonstrated below, ensure that **Track** using all polar angle settings is enabled. Then, draw a circle with a radius of 0.5 and use OSNAP, OTRACK, and polar tracking to specify its center.



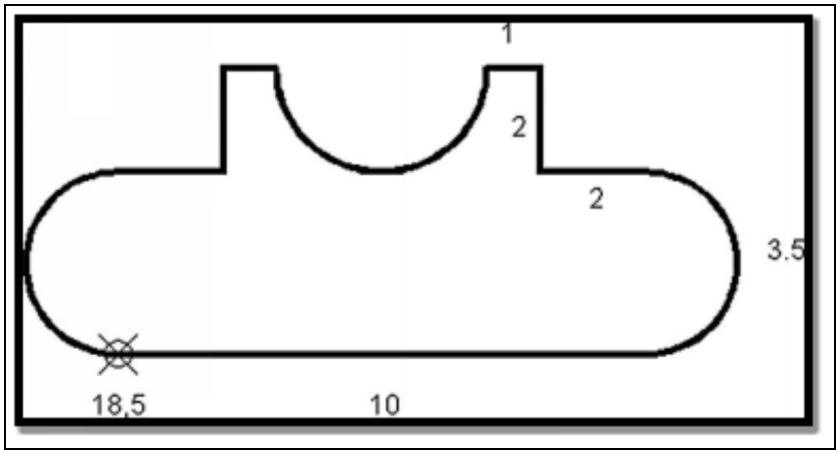

- Follow these steps to create the final shape, which is a circle at the top:



- Close and save the file.

# CONVERTING AND DRAWING POLYLINES

- Launch AutoCAD 2025 and select Practice 2-5.dwg.
- With a width of 0.1 and a start point of 18,5, draw the following polyline.

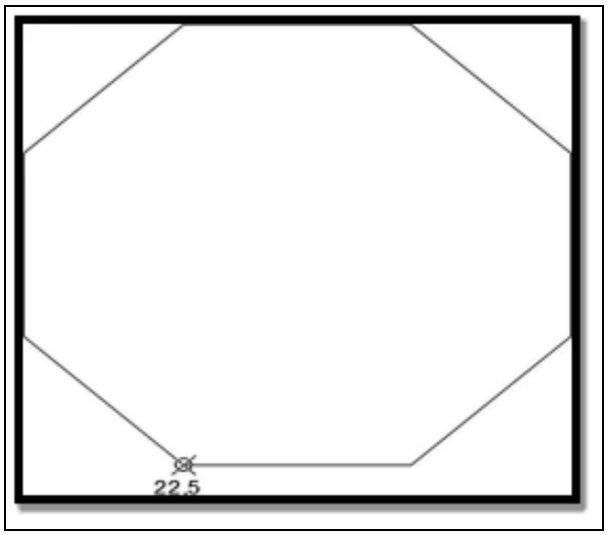



- Next, let the polyline burst. The breadth will vanish.
- After exploding, check the objects; they are now lines and arcs.
- Proceed to save and close.

# GRID AND SNAP

- Launch **AutoCAD 2025** and select Practice 2-6.dwg.
- To use angle = 45 in Polar Tracking, change it.

- Set the distance to 0.5 and switch the **Snap to Polar Snap**.
- Draw the following form, using an angle of 45 and starting from point 22,5, with all segment lengths equal to 6.5:




- Close and save the file.

# CHAPTER THREE

# ADJUSTING COMMANDS

## Selecting Objects in AutoCAD 2025

Before using any of the Modifying commands covered in this chapter, it is essential to begin by selecting the "**desired objects**". After executing any of the modifying commands, a prompt will be displayed:

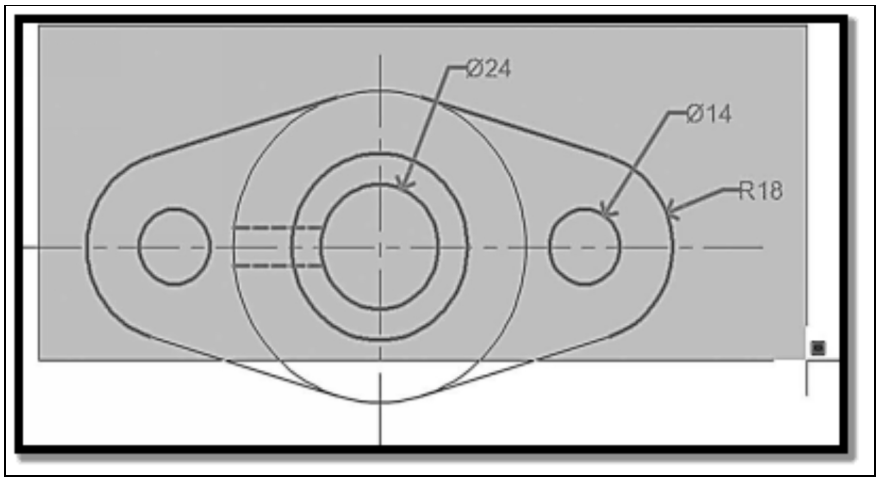
## Select objects

When you select an object, the cursor will transform into the Pick Box. When using this prompt, you have the option to work without any keyboard input or simply type a few letters to activate a specific mode.

**Perform a variety of tasks without the need for manual input:**

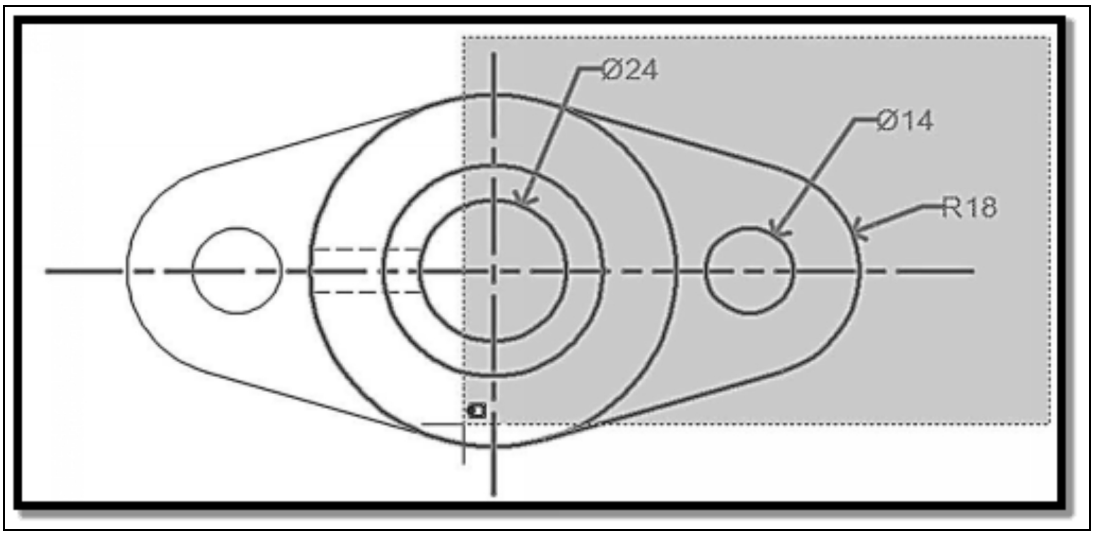
- When selecting objects in AutoCAD, you can simply click on them using the pick box. Take your time and select each object one by one.
- To initiate Window mode and select all objects fully contained within the window, simply click and drag to the right without selecting any object beforehand.
- To initiate Crossing mode, simply click and drag to the left without selecting any object. This mode will then select all objects that are fully contained within the Crossing or are touched (crossed) by it.

**Here are two examples for you to review: Example of a window:**




You will notice a small rectangle with a blue circle inside it at the cursor. Only objects other than the large circle and the diagonal lines at the bottom will be selected. Can you please explain the reason behind this? These elements are not completely contained within the window.

**Example of a crossing:**




You will notice a small rectangle intersecting the green circle at the cursor. Only the two diagonal lines at the left, along with the arc and circle, will not be selected. Activating a specific mode in the Select Objects prompt can be done by simply typing a few letters. Here are the letters you can use:

# Crossing Mode (C)

By entering C at the command prompt, you can effortlessly switch the selecting mode to the Crossing. This handy feature allows you to select objects whether they are to the right or left of your current position.

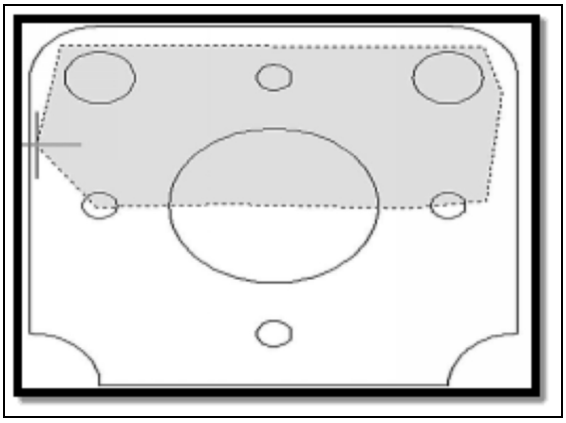
# Window Mode (W)

When you're working in AutoCAD, you can easily switch the selecting mode to the Window by typing W at the command prompt. This mode allows you to select objects by drawing a window around them, whether you're moving from right to left or vice versa.

# Crossing Polygon Mode (CP)

Given the existence of W and WP, it logically follows that there would also be C and CP. Using Crossing Polygon in AutoCAD allows you to define a shape that is not a rectangle to enclose and intersect objects.

**Take a look at the illustration below:**



Six circles will be chosen based on their positioning to the CP. Three circles will be fully contained within the CP, while the other three will be intersected by the CP.

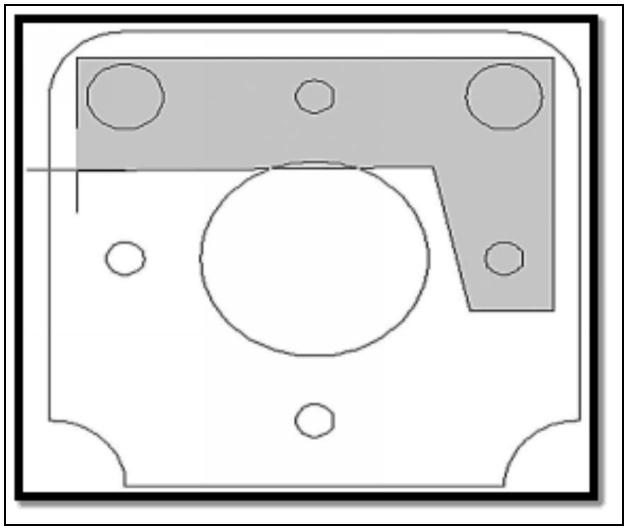
# Window Polygon Mode (WP)

In WP mode, you have the flexibility to specify points in any manner you prefer, allowing you to define a non-rectangular window. When you type WP and press [Enter], you will be presented with the following prompts:

**Specify the first point of the polygon:**

- Specify endpoint of line or [**Undo**]:
- Specify endpoint of line or [**Undo**]:
- Press the [**Enter**] key to exit WP mode. When working with WP, it is important to ensure that the object is fully contained within it to select it accurately.

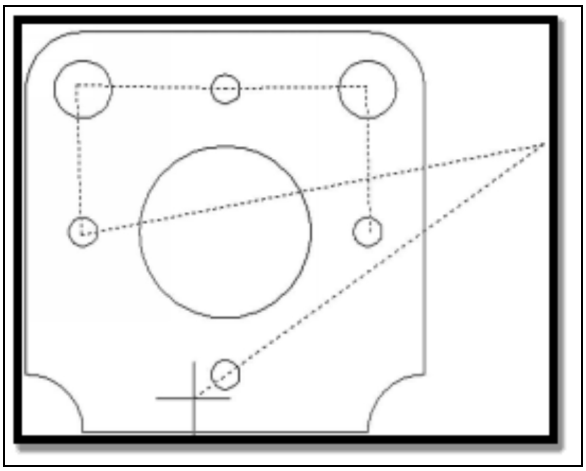
**Take a look at this example:**



Only four circles will be chosen, all of which are completely enclosed within the WP.

# Fence Mode (F)

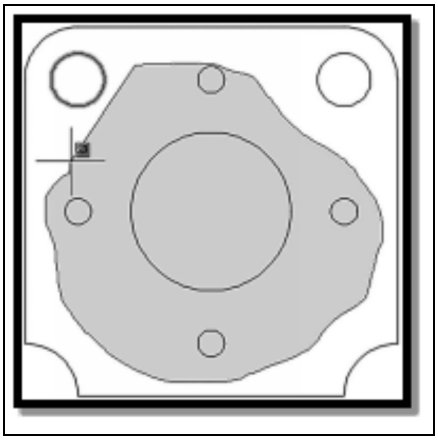
In Fence mode, you can conveniently select multiple objects by crossing or touching them. **Lines in Fence mode can intersect each other, which is different from WP and CP. Take a look at the example below:**



All the circles have been selected as they are intersected by the fence. The vertical line on the right will also be selected. It is perfectly acceptable in AutoCAD to have crossed fence lines.

# Lasso Selection

You can seamlessly combine the WP and CP modes without the need for any manual input. When the pick box is visible, simply click and hold. Moving to the right will give you WP while moving to the left will give you CP. **Keep in mind that the shape may be irregular, similar to the example below:**



# Previous (P), Last (L), and more modes

**In addition, objects can be selected using various modes:**

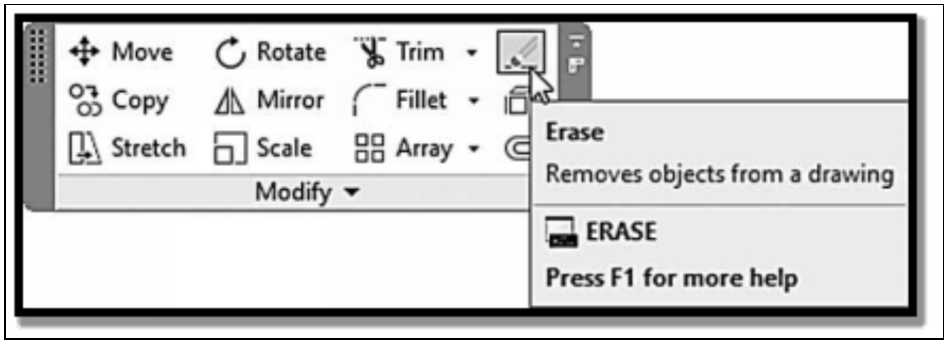
- **Last (L):** Select the most recently drawn object using the Last (L) command.

- **Previous (P):** To select the most recently selected objects, use the "**Previous**" command (P).
- **All**: This command allows you to quickly select all objects in the drawing.

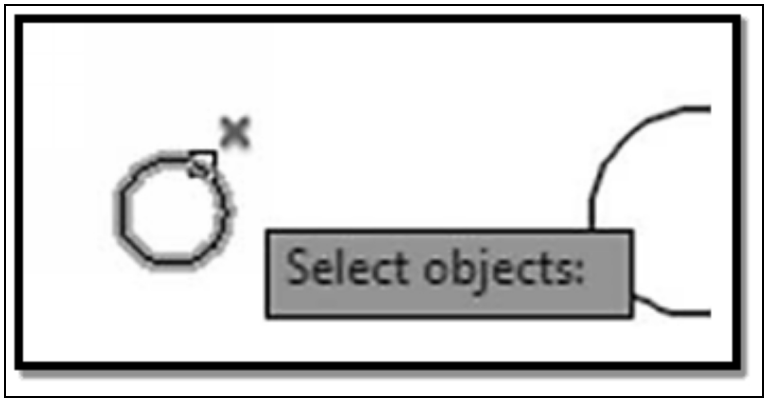
To deselect objects, simply hold down the [**Shift**] key on your keyboard and click on the desired objects. When operating in this mode, you can use the window and crossing functions. Initiate a selection window in a specific area of your drawing, and seamlessly navigate to another section while ensuring that the objects outside the visible screen are still selected.

# How to Use Erase Command

The Erase command is capable of deleting any object that you choose. To execute the command, navigate to the **Home tab**, find the **Modify panel**, and click on the **Erase button**:

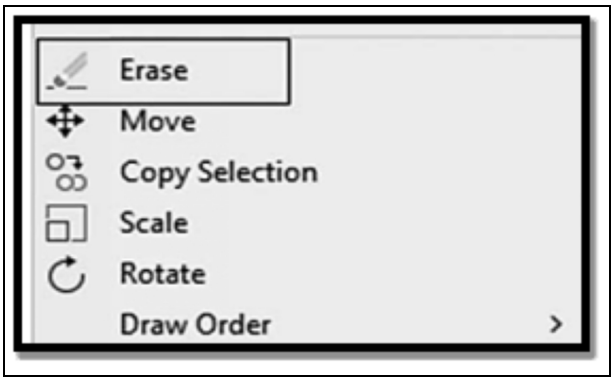



**After opening the command window, a prompt will be displayed:**
**Choose the desired objects:**
**When using the one-by-one method, you will observe a transformation in the cursor's appearance:**



Choose the objects you want. To conclude the **Select Objects prompt**, you must either press [**Enter**] or right-click, as it tends to repeat itself.
**There are alternative methods available for erasing:**

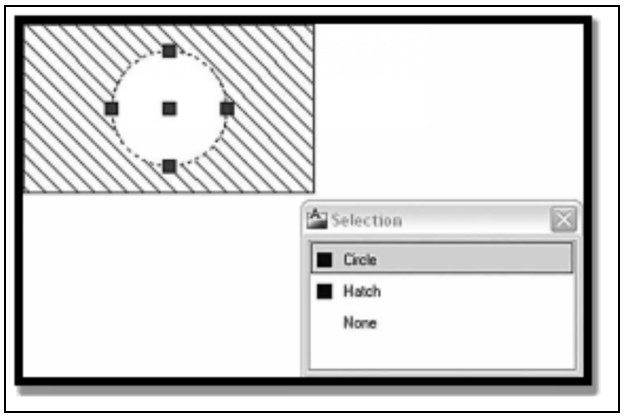
- To delete the desired object(s), simply select them and press the [**Del**] key on your keyboard.
- To delete an object in AutoCAD, simply select the desired object(s), right-click, and choose the Erase option from the shortcut menu.



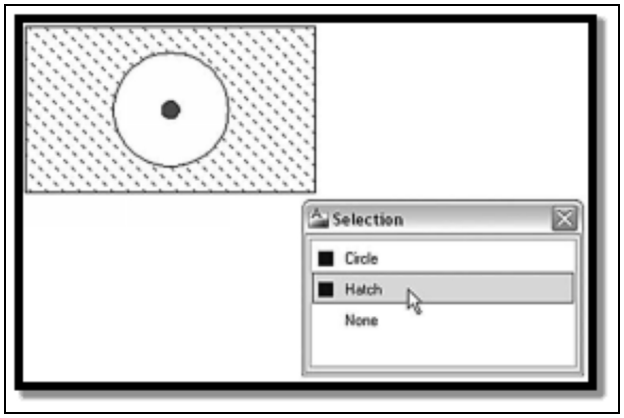

# How to Use Selection Cycling

When working with AutoCAD, it's important to be mindful of potential issues that may arise during the drafting process. One common mistake is accidentally overlapping objects or selecting a point that is shared by multiple objects. This used to be a concern in previous versions, but it has been resolved in the current version. Selection Cycling in AutoCAD is a useful feature that alerts us when our click intersects multiple objects. It empowers us to easily select the desired object with precision. To activate Selection Cycling, you can easily access the feature by clicking on the Selection Cycling button located on the Status bar. If the button is not visible, simply click on the last button at the right side of the Status bar, which is represented by three horizontal dashes. This will expand the button options, allowing you to select the Selection Cycling feature.

**After activation, when selecting an object, the following may be displayed:**

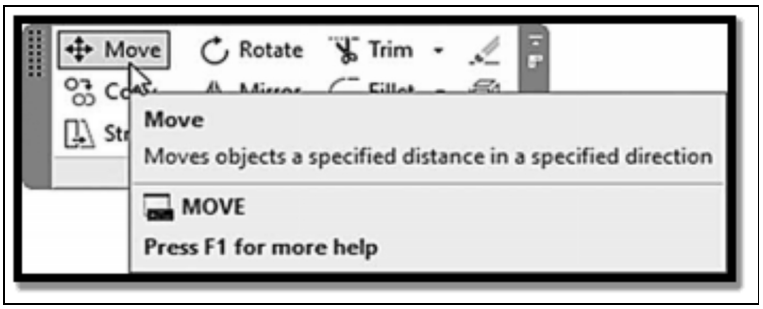



The window will display two potential objects at the clicked location, namely Circle and Hatch, and it will indicate that Circle has been chosen. **By navigating to the Hatch option in this window, you will be presented with the corresponding image:**

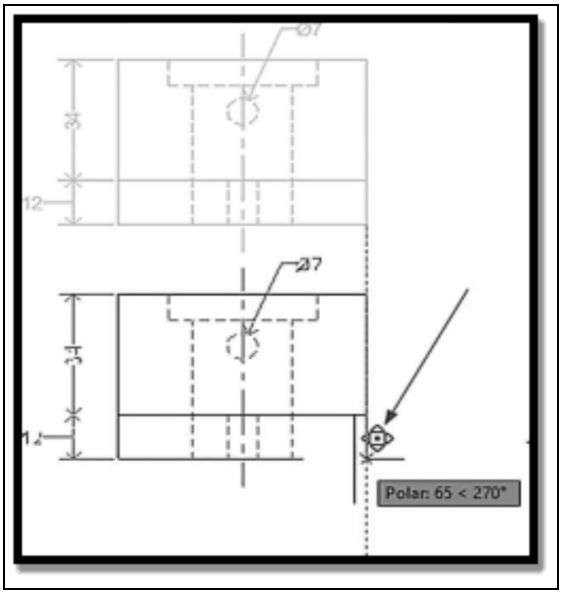


# How to Use Move Command

The move command allows for the relocation of objects within the drawing. To execute this command, navigate to the **Home tab**, find the Modify panel, and click on the **Move button**:



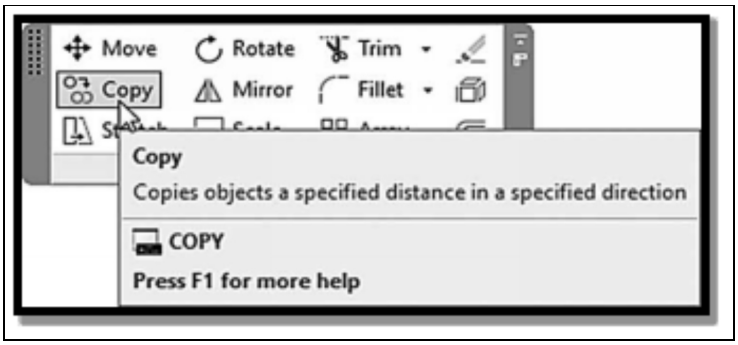

**Here are the prompts that will be displayed:**
Select objects:
Specify base point or [Displacement] <Displacement>:
Specify second point or <use first point as displacement>:
To begin, objects must be selected, followed by the selection of the Base Point. The base point serves as the reference point for the objects, allowing them to be moved at a specific distance and angle. The primary focus of the Basepoint is to ensure precision.
Once the Basepoint is selected, the cursor shape will transform the following:



For the final prompt, you'll need to indicate the second point or destination for your movement.

# How to Copy Command

To copy objects, you can use this command. To execute this command, navigate to the Home tab, find the **Modify panel**, and click on the **Copy button**:




**After launching the program, a series of prompts will appear:**
Select objects:
Current settings: Copy mode = Multiple
Specify base point or [Displacement/mode] <Displacement>:
Specify second point or [Array] <use first point as displacement>:
Specify second point or [Array/Exit/Undo] <Exit>:
Specify second point or [Array/Exit/Undo] <Exit>:
Once you have chosen the objects you want, AutoCAD will provide you with the current mode, which in this case is Multiple. This feature allows users to easily generate multiple copies within a single command. Another mode available is a Single copy. When prompted, you will be asked to specify the base point. After selecting the first point, AutoCAD will prompt you to specify the second point to complete the copy process. It will then repeat this prompt for creating additional copies if desired.
**There are three options available for use:**

- To undo the last copy, you can use the **Undo command**.
- Use the "**Exit**" command to conclude the task.
- Use the **array command** to effortlessly create multiple copies of an object with precise spacing and rotation. When you choose

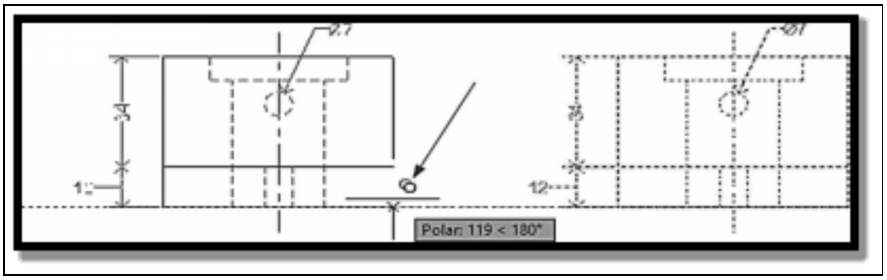
the array option, a series of prompts will appear:

Enter number of items to array:

Specify second point or [Fit]:

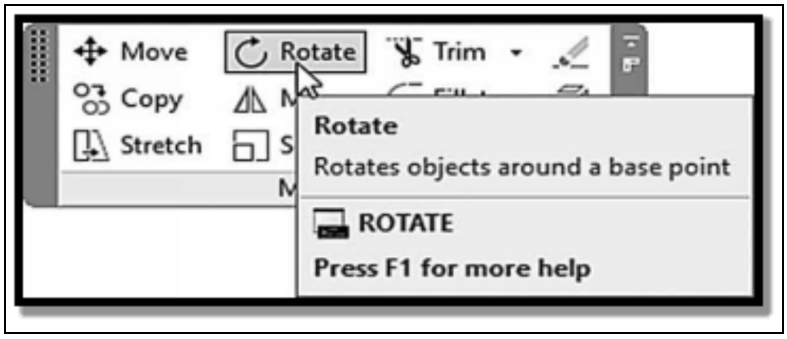
Provide the total number of items in the array, including the original object. Additionally, indicate the distance between each of the objects. You can use the Fit option to define the overall distance, and AutoCAD will evenly distribute the distance across the objects.

**Once the Basepoint is selected, the cursor shape will transform the following:**




# How to Use Rotate Command

This command allows for the rotation of objects around a specified base point, using either a specific rotation angle or a reference point. To execute the command, navigate to the Home tab, find the **Modify panel,** and click on the **Rotate button**/




After launching the program, a series of prompts will be displayed for your attention:

Current positive angle in UCS: ANGDIR=counterclockwise ANGBASE=0

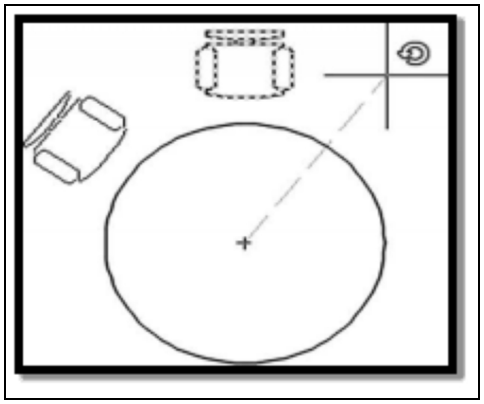
Select objects:

Select objects:

Specify base point:

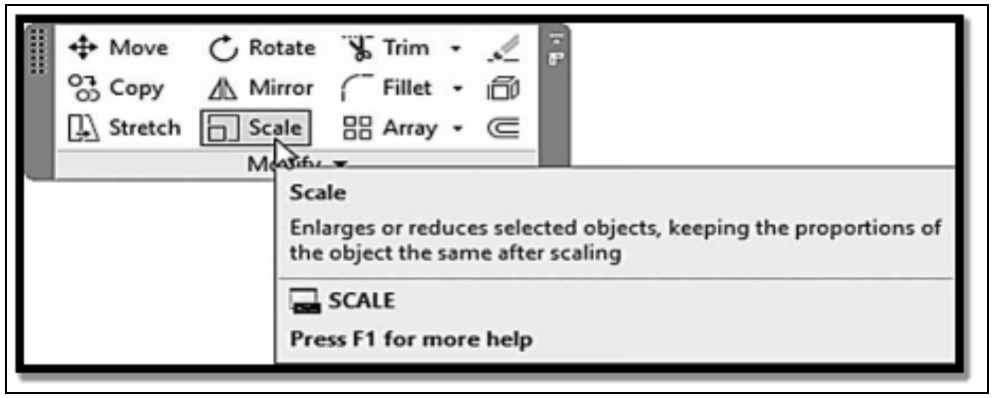
Specify rotation angle or [Copy/Reference] <0>:

One important message is to inform you about the current angle direction and the Angle Base value. In this case, the rotation point serves as the base point for all the selected objects, causing them to rotate around it.
Use the **Copy function** to duplicate the chosen elements, followed by a rotation.
Once the Basepoint is selected, the cursor shape will transform, assuming the following appearance:



# How to Use Scale Command

This command allows you to easily resize objects by specifying a scale factor or a reference point. To execute this command, navigate to the **Home tab**, find the **Modify panel,** and click on the **Scale button**:




After launching the program, a series of prompts will be displayed:

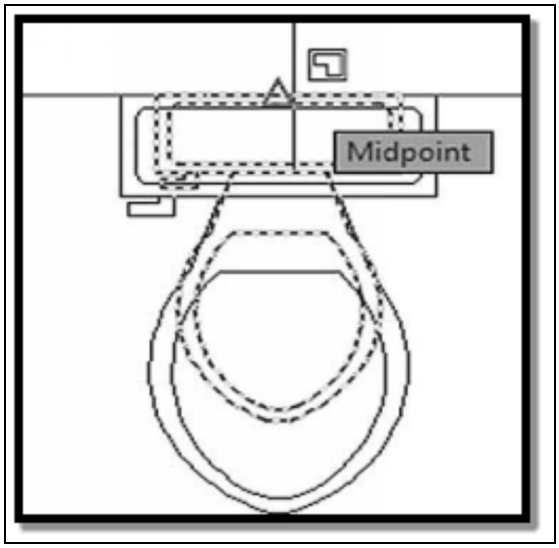
Select objects:

Specify base point:

Specify scale factor or [Copy/Reference] <1.0000>:

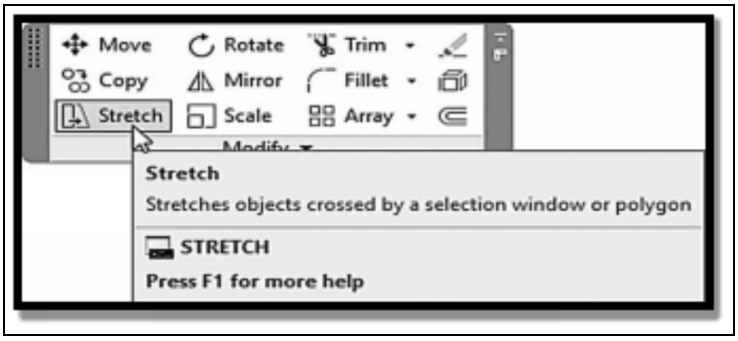
In this case, the scaling point serves as the base point for resizing the selected objects. They will either increase or decrease in size to this point. Use the **Copy function** to duplicate the chosen elements, and subsequently adjust their scale.

**Once the Basepoint is selected, the cursor shape will transform the following:**




# How to Use Strech Command

This command allows you to modify the length of selected objects by stretching them based on a specified distance and angle. To execute this command, navigate to the Home tab, find the Modify panel, and click on the **Stretch button**.



**After opening the program, a series of prompts will be displayed:**

Select objects to stretch by crossing-window or

crossing-polygon...

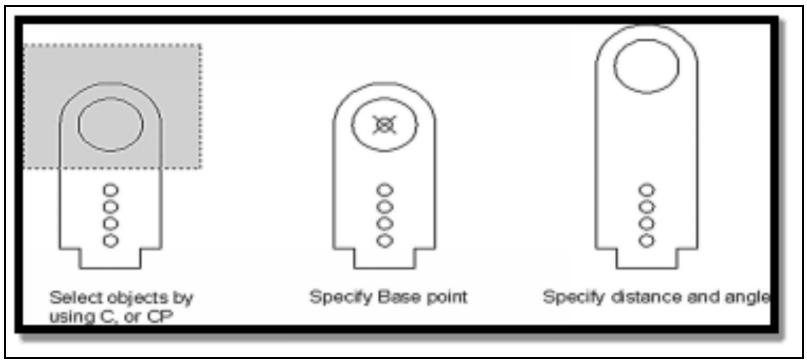
Select objects:

Specify base point or [Displacement] <Displacement>:

Specify second point or <use first point as displacement>:

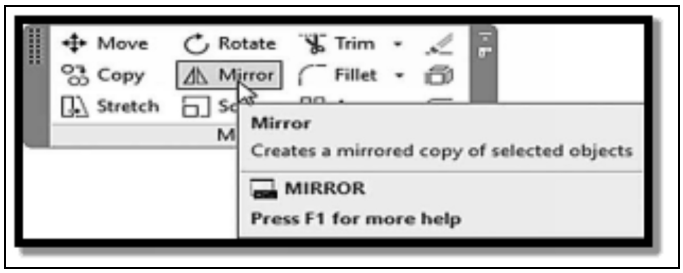
The Stretch command stands out from other modifying commands due to its unique approach of selecting objects using C or CP modes. What is the reason behind this? Stretch will utilize both the containing and crossing features of C and CP. Objects that are completely contained within the C or CP will be in motion, while objects that intersect with these boundaries will stretch by either increasing or decreasing in length. Next, you'll need to indicate the base point (similar to the Move and Copy commands) and then specify the second point.

**Take a look at the following example:**




# How to Use Mirror Command

To create a mirror image of the selected objects, you can utilize the mirror line command. To execute this command, navigate to the **Home tab**, find the Modify panel, and click on the **Mirror button**.




**After launching the program, a prompt will be displayed:**

Select objects:

Specify first point of mirror line:

Specify second point of mirror line:

Erase source objects? [Yes/No] <N>:

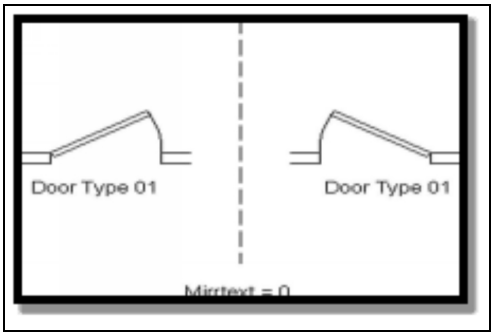
Once you have selected the objects, you can easily define the mirror line by specifying two points. It is not necessary to draw a line to specify these two points. In the final prompt, you will be given the option to either retain or remove the original objects.

Text can be included in the selection set for mirroring. AutoCAD allows you to specify the desired action (copying or mirroring) by using the system variable **MIRRTEXT**. To execute this command, enter it into the command window, and the following will be displayed:

Enter new value for MIRRTEXT <0>:

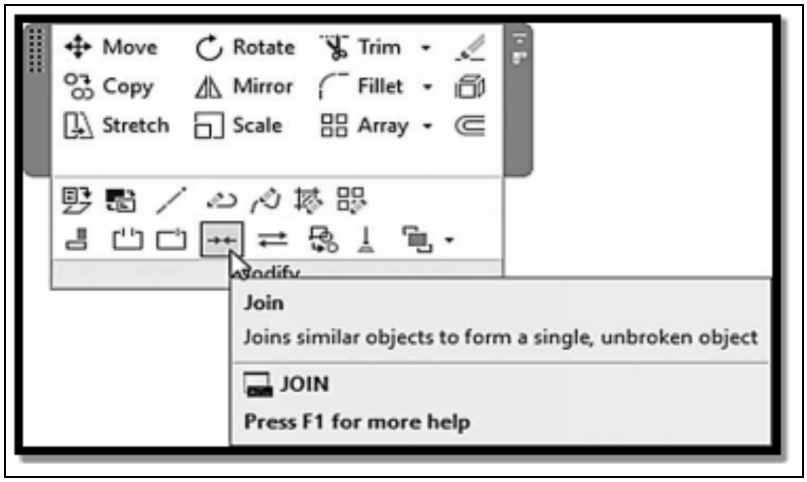
You have the option to input either 1 or 0. Choosing 1 will mirror the text to match the other objects while choosing 0 will create a copy of the text.

**Take a look at the following example:**



# How to Join Objects

This command is extremely useful as it allows you to effortlessly connect lines, arcs, and polylines. The Join command simplifies and streamlines the process, making it easy to accomplish all of the tasks mentioned. To execute this command, navigate to the **Home tab**, find the **Modify panel**, and click on the **Join button**:




**When using AutoCAD, you will be presented with the following prompts:**

Select source object or multiple objects to join at once:

Select lines to join to source:

Select lines to join to source:

1 line joined to source

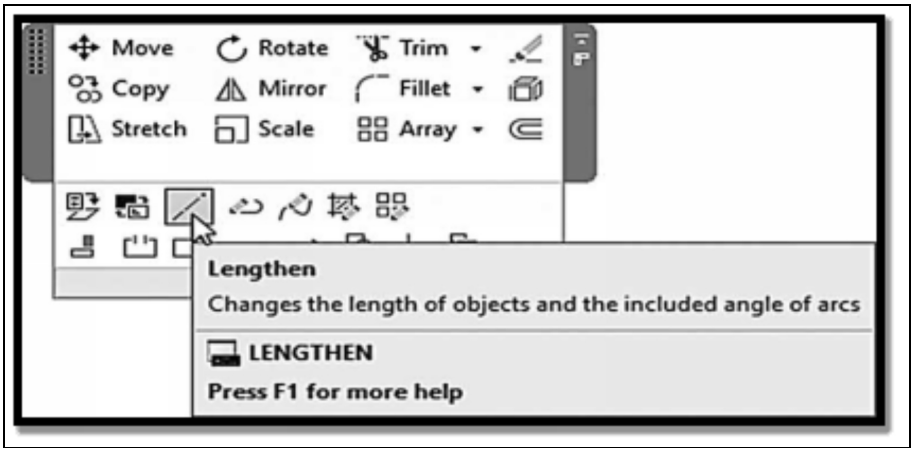
The prompts provided above are specific to the process of joining lines. It is important to note that the prompts may vary when dealing with arcs and polylines. For the joining process to be successful, certain conditions must be met:

- When combining lines, arcs, and polylines to create a single polyline, it is important to ensure that the ends of the objects are properly connected.
- When the lines, arcs, and polylines are not connected, AutoCAD treats each connected group as a single polyline.

- When joining lines to form a line, it is important to ensure that they are always collinear.
- When joining arcs to create a single arc, it is important to ensure that they share the same center point.
- When joining arcs in AutoCAD, you will come across a specific prompt that asks if you want to create a circle. This command is unique as it is the only one that can create a circle from an arc.

# How to Lengthen Objects

This command allows you to modify the length of objects by either adding or subtracting length using various methods. To execute this command, navigate to the Home tab, find the **Modify panel,** and click on the **Lengthen button**:




**Here is the prompt that will be displayed to the user:**

Select an object or [DElta/Percent/Total/DYnamic]:

By selecting an object, AutoCAD will provide you with the precise measurement in the units you have set. There are various techniques available in AutoCAD to adjust the length of objects.

One method available is Delta, which enables users to easily adjust the current length by adding or removing. When working with values, it's important to understand that positive values indicate addition, while negative values indicate subtraction.

Here's what you'll see when you select the Delta option:

Enter delta length or [Angle] <0.0000>:

Another option is the Percentage method, which allows users to adjust the length by specifying a percentage of the current length. To increase the length, enter a value greater than 100. Conversely, to decrease the length, enter a value less than 100.
Here's what you'll see when you select the Percentage option:

Enter percentage length <100.0000>:

Another option available is the Total method, which allows the user to input a new total length for the object. When the new value exceeds the current value, the object will be extended by the specified length. Conversely, if the new value is smaller, the length will be subtracted from the object.
**Here's what you'll see when selecting the Total option:**

Specify total length or [Angle] <1.0000)>:

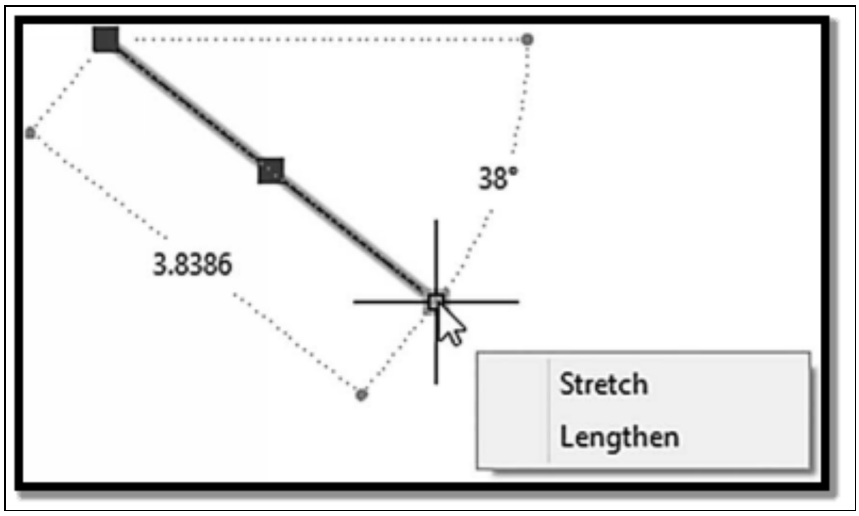
The fourth method is Dynamic, allowing for dynamic length adjustments using the mouse. Upon opening the program, a prompt will appear:

Specify new endpoint:

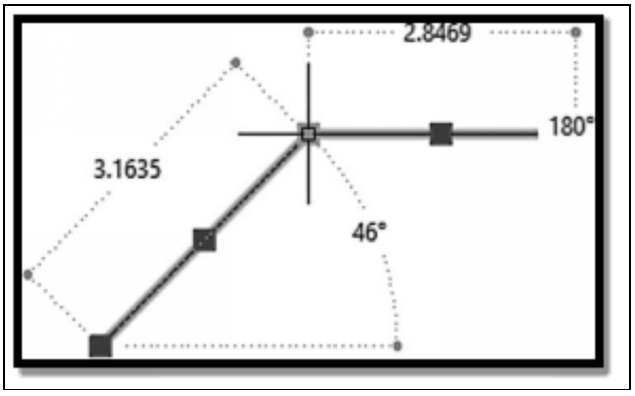
**Take note of the following:**
When working in AutoCAD, it is important to remember that you can adjust the length of a single object using just one method per command. This efficient approach allows you to quickly and accurately modify the size of your objects without any unnecessary complications.

# How to Use Grips and Dynamic Input

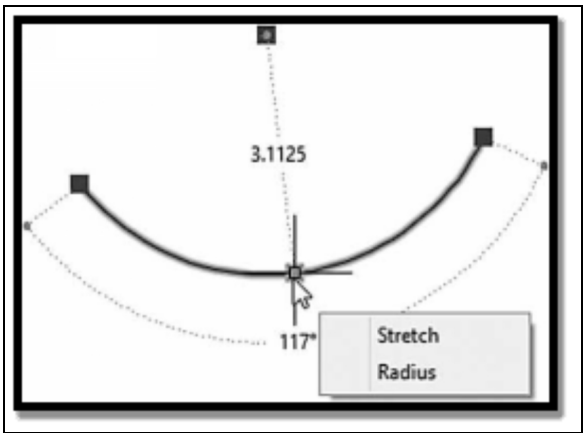
When you hover over a grip without clicking, Dynamic Input will provide you with valuable information about the selected objects, tailored to their specific type. Ensure that Dynamic Input is enabled in the Status bar.
**Take a look at the following examples:**
When utilizing the line tool and selecting one of the endpoints, you will be able to view the length and angle to the east. Additionally, AutoCAD provides two useful commands, Stretch and Lengthen, to further manipulate your lines.



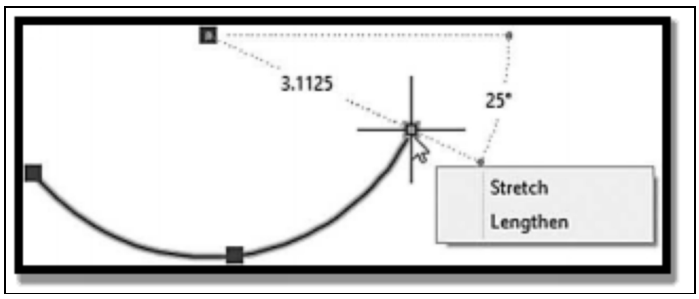

When the two lines are connected, they will display the respective lengths and angles.



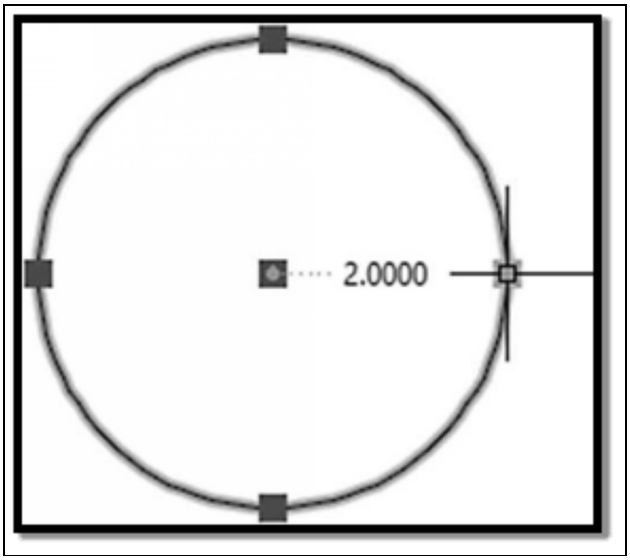

When dealing with an arc, you'll be able to easily determine the midpoint by utilizing two important commands: Stretch and Radius. These commands will not only display the radius and included angle, but also provide you with the necessary tools to manipulate the arc as needed.




In AutoCAD, the endpoints of an arc display the radius and angle for the east. You can use the Stretch and Lengthen commands to modify the arc.

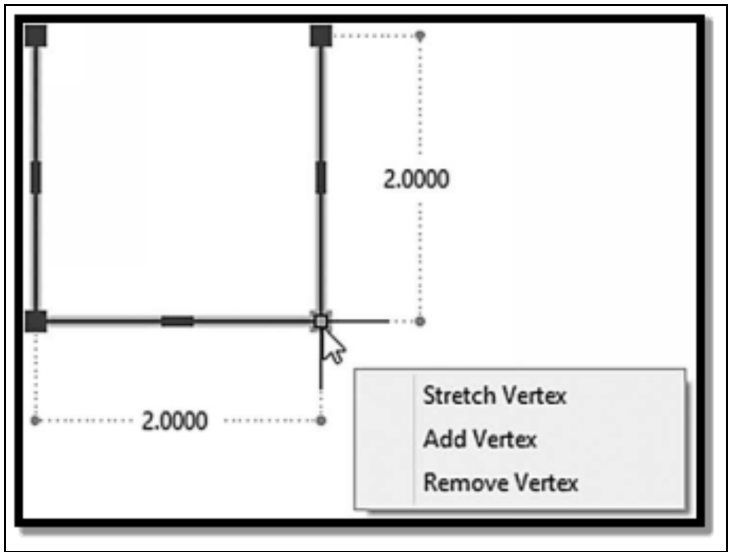



When it comes to a circle, the only thing you'll need to consider is the radius.




When two segments of a polyline share endpoints, you will be able to see the length of the shared lines. Additionally, a convenient shortcut menu will

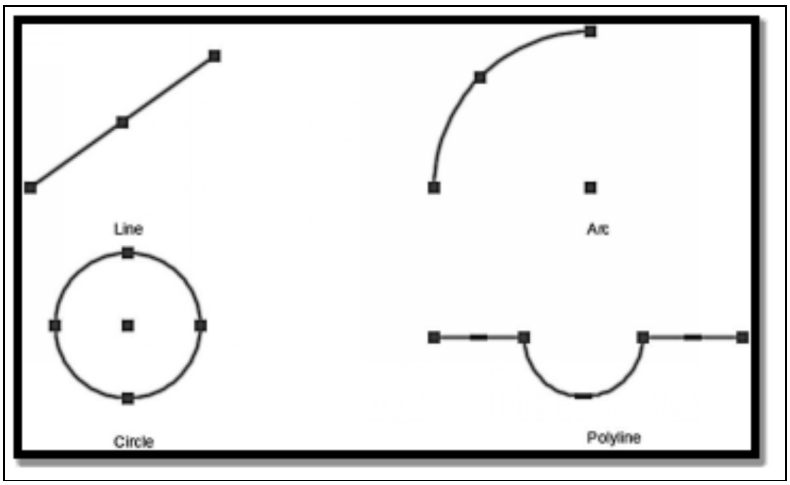
appear, offering options such as Stretch Vertex, Add Vertex, and Remove Vertex.




# How to Use Grips to Edit Objects

Grips are visual indicators in the form of blue squares and rectangles that are strategically placed on various objects within AutoCAD. You can easily modify these objects using five different modifying commands, using the selected grip as a base point. AutoCAD is an incredibly efficient tool that allows for faster modifications without sacrificing accuracy. It surpasses the limitations of traditional modifying commands.

**Based on the object type, grips will be displayed at various locations, as shown in the accompanying illustration:**



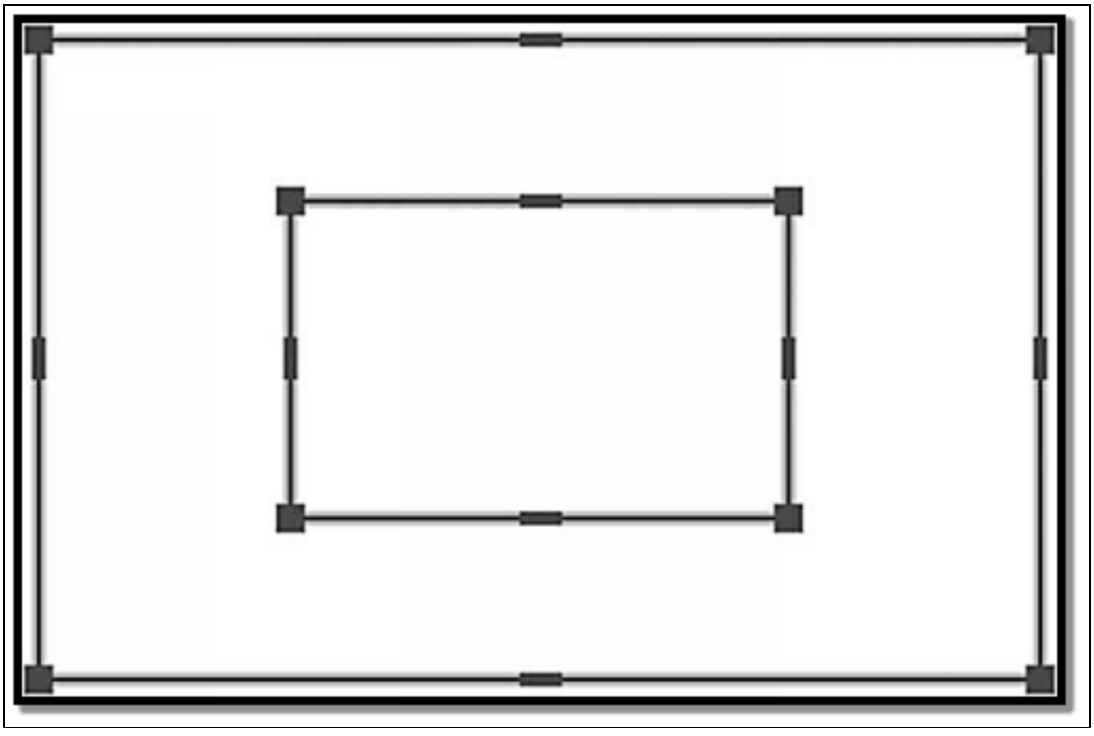

Typically, the grip color is set to blue, representing a cold temperature. However, when you click on it, the color changes to red, indicating a hot temperature. When you right-click on an object and select the appropriate grip, a set of modifying commands will appear that all have one thing in common: the base point. When you select the grip located at the end of a line **and perform a right-click, a menu will appear with the following options:**

**Six commands can be observed:**

- Lengthen

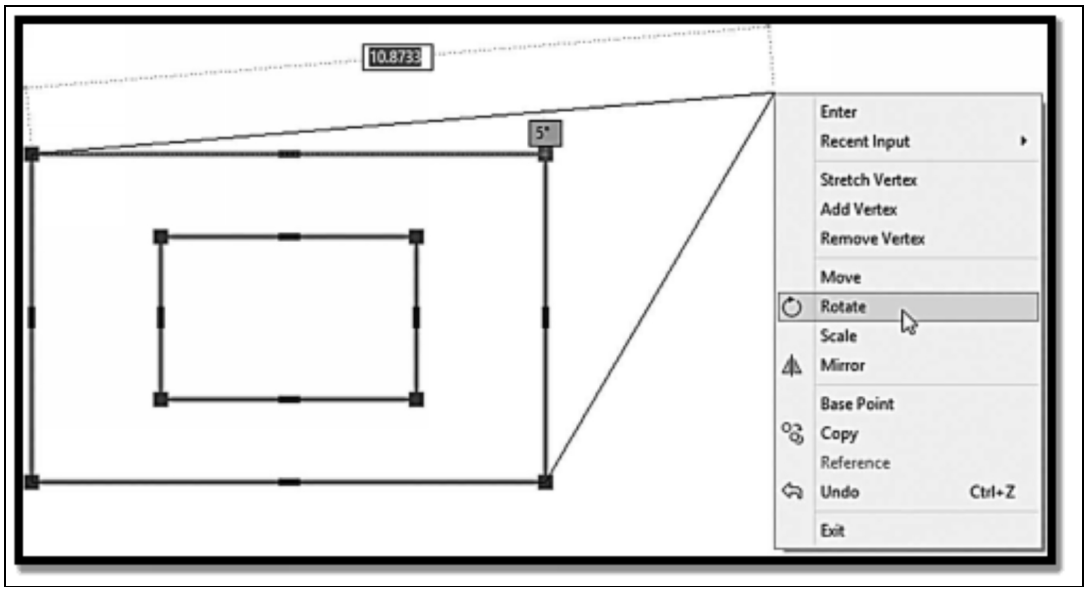
- Stretch
- Scale
- Mirror
- Move
- Rotate

All of these commands, except for Lengthen, have a common base point. This means that if you treat the first point of the mirror line as a base point, it will be equivalent to the others. Copying in AutoCAD is a mode that seamlessly integrates with all other commands, allowing for efficient workflow. To select multiple base points, hold down the [**Shift**] key while selecting the grip. When you hold down [**Ctrl**] while performing certain actions in AutoCAD, such as specifying a second point for Move and Stretch, setting a rotation angle for Rotate, specifying a scale factor for Scale, or selecting a second point for Mirror, the software will remember your last input and allow you to repeat it graphically. You have the option to select a different base point instead of the grip that was initially selected. You can easily select the desired shape by choosing the two polylines.

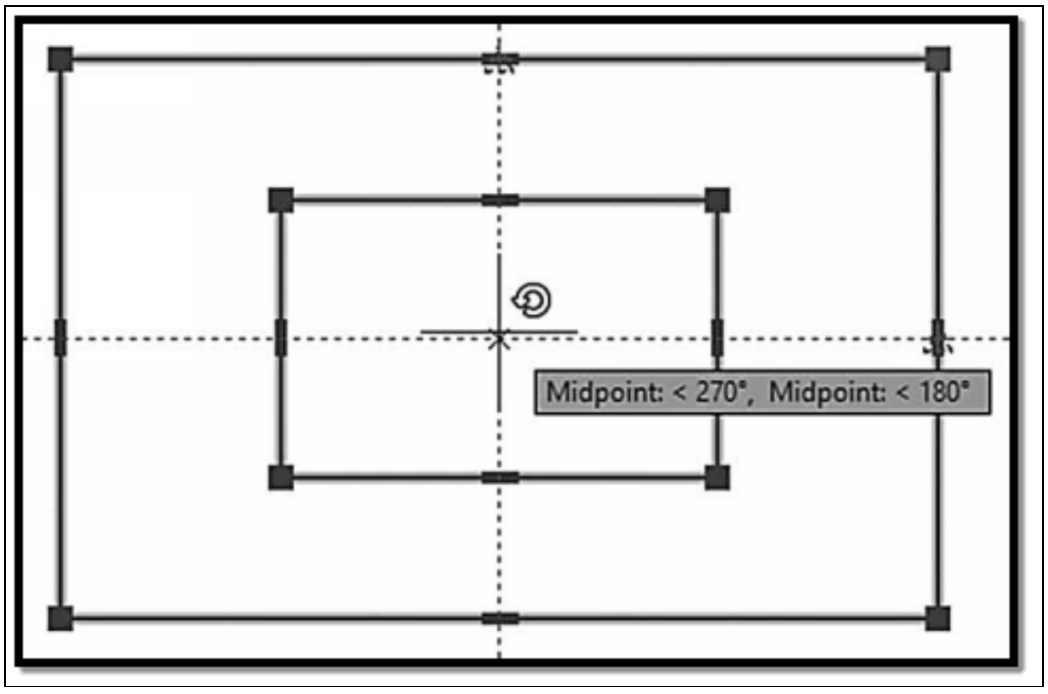


To activate the desired function, simply click on the top right corner. To access additional options, right-click and a menu will appear. From the

menu, choose the **Rotate command** to proceed.

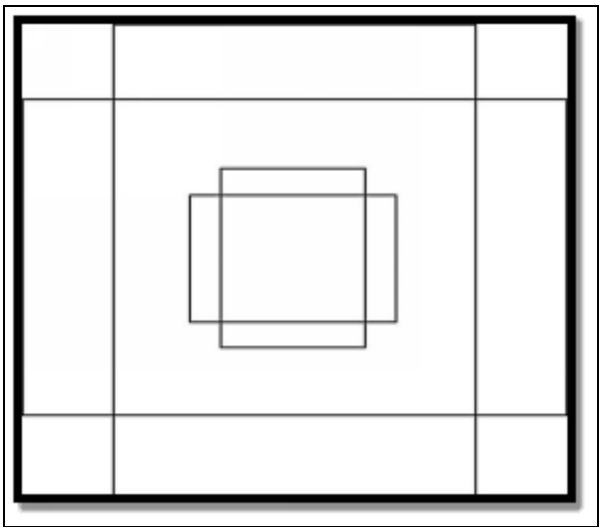



To rotate the two shapes, simply use the grip as your base point. Right-click and choose the Basepoint option to select another base point. In this case, the center of the two rectangles can be selected using OSNAP and OTRACK.

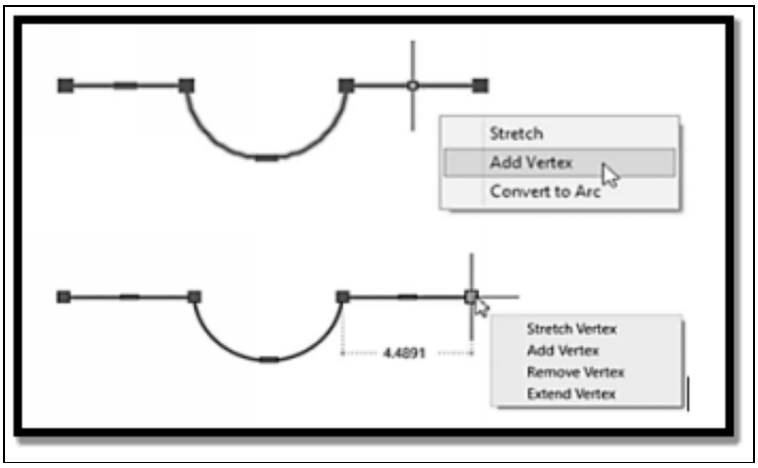



To copy while rotating, simply right-click once more and choose the **Copy option**. Use Polar tracking to set an angle of 90 degrees, and then press the

[**Esc**] key. Here is the result you will receive:



When working with certain objects such as polylines in AutoCAD, you will notice that they display more than just blue squares. They will also show rectangles positioned at the midpoint of each line and arc segment. The rectangles located at the midpoint of the polyline serve various purposes. AutoCAD displays a distinct menu based on whether the segment is a line or an arc. **Hover over the grip without clicking, and you'll observe the following:**




To manipulate the middle grip, you have a few options at your disposal. You can stretch it, add a vertex, or convert it to an arc (if it was originally a line) and convert it back to a line (if it was originally an arc). To achieve the desired grip at the end, you have several options in AutoCAD. You can use the Stretch Vertex, Add Vertex, Remove Vertex, or Extend Vertex

commands. The Extend Vertex command allows you to keep the last segment and add a new segment, providing flexibility in your design. After completing the necessary adjustments with grips, simply press the [**Esc**] key once or twice, depending on the specific context, to exit the grips mode.

# CHAPTER FOUR

# ADJUSTING COMMANDS CONT'D

These commands have advanced capabilities for modifying objects. They can construct on top of the shapes you have created. Every command in AutoCAD serves a distinct purpose. Some commands, such as Offset and Array, are used for creating objects, while others like Fillet, Chamfer, Trim, and Extend, are designed to modify the shape of existing objects.

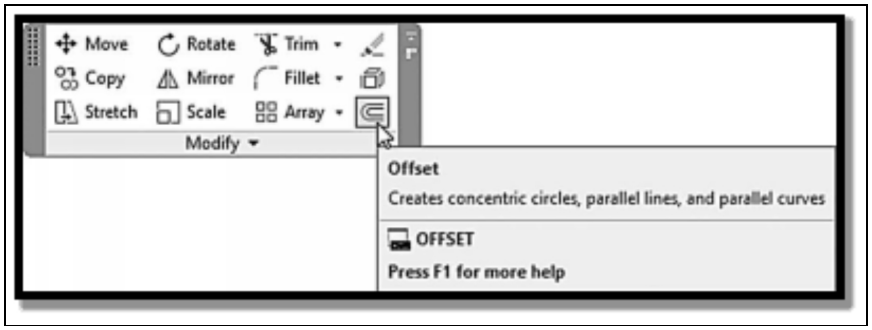
**Here is a concise overview of each command:**

- **Offset command**: The Offset command is used to create parallel copies of an object.
- **Fillet command:** The Fillet command is used to create a clean intersection between two objects, whether it's by extending or trimming lines or using arcs.
- **Chamfer command**: The Chamfer command is used to create a clean intersection, but it is limited to working with lines. To create a clean intersection, you can extend/trim the two lines or create a new line to represent the chamfered edge.
- **Trim command:** The Trim command is a powerful tool that allows you to precisely trim objects by using other objects as cutting edges.
- **Extend command:** The Extend command in AutoCAD allows you to extend objects by using other objects as boundary edges.
- **Array command**: The Array command in AutoCAD allows you to create objects using three different methods: rectangular, circular, and using a path.
- **Break command:** The Break command is used to split an object into two separate objects by selecting two points and removing the portion between them.

## About Offsetting Objects

The Offset command is used to generate duplicates of an object that are aligned with the original. The new object will have identical properties to the original object. Offsetting can be achieved by specifying the offset

distance or by selecting a point through which the new object will pass. To initiate this command, navigate to the **Home tab**, locate the **Modify panel**, and click on the **Offset button**:




**Here are the AutoCAD prompts that will be displayed to the user:**

Current settings: Erase source=No Layer=Source

OFFSETGAPTYPE=0

Specify offset distance or [Through/Erase/Layer] <Through>:

# How to Offset Using the Through Option

If you are unsure about the offset distance, but have a specific point in the drawing that the new parallel object needs to pass through, this option will assist you in achieving your goal.
These are the prompts you will encounter in AutoCAD:

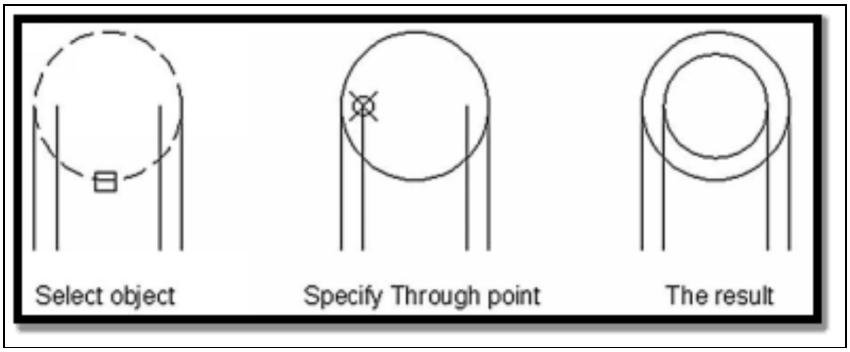
Specify offset distance or [Through/Erase/Layer] <Through>:

Select object to offset or [Exit/Undo] <Exit>:

Specify through point or [Exit/Multiple/Undo] <Exit>:

With this feature, you can easily generate a single offset. To create additional offsets using the offset command, simply select another object and repeat the same steps. Pressing the [**Enter**] key or right-clicking will terminate the command.

**Here’s an exemplary demonstration:**




# How to Offset Using the Offset Distance Option

To specify the distance between the object and the new parallel copy, enter the desired value. Next, select the original object and click on the desired side for the placement of the new object. By following this approach, you will encounter the following prompt:

Specify offset distance or [Through/Erase/Layer] <Through>:

Select object to offset or [Exit/Undo] <Exit>:

Specify point on side to offset or [Exit/Multiple/ Undo] <Exit>:

With this feature, you'll be able to effortlessly generate a single offset. To create additional offsets using the offset command, simply select another object and repeat the same steps. Pressing the [**Enter**] key or right-clicking will complete the command.

# How to Use the Multiple Option

With AutoCAD, you have the option to easily repeat the Offset Distance or Through option within the same command. Simply click on the side of the offset or specify a new through point. Here are the prompts for Multiple:

Specify through point or [Exit/Multiple/Undo] <Exit>:

**M** Specify point on side to offset or [Exit/Undo] <next object>:

**When offsetting, it is important to keep the following in mind:**

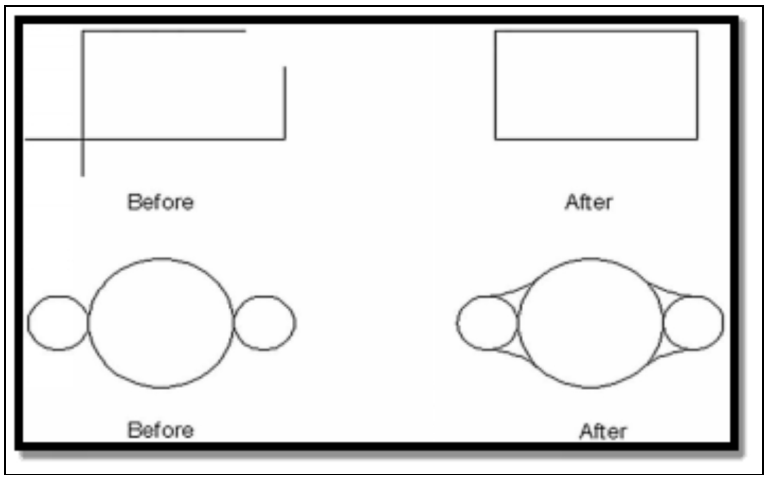
- If any mistakes are made, the Undo option can be used.
- AutoCAD can remember and save the last offset distance used in the file.
- When offsetting an arc or circle, the new arc and circle will have a common center point. This will result in a smaller or larger arc

or circle.

- When offsetting a closed polyline, the resulting output can either be smaller or larger.
- The Offset tool in AutoCAD includes a convenient automatic preview feature that allows you to see the result before confirming your selection.

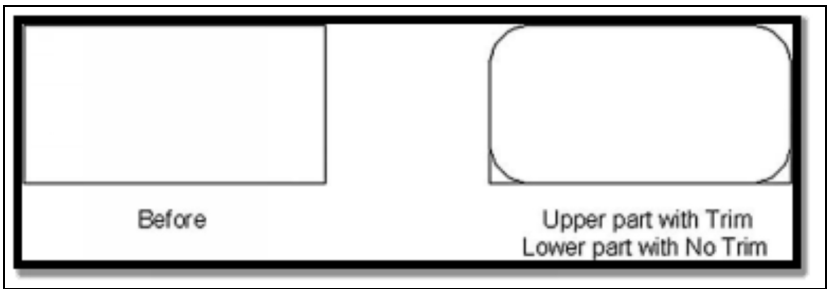
# How to Fillet Objects

The purpose of the Fillet command is to generate clean intersections. As a first step, it is important to set the value of the Radius. When the value is 0, you have the option to utilize Fillet exclusively between two lines. Fillet will then proceed to extend or trim the lines to the suggested intersection point. If the Radius value is greater than 0, Fillet can utilize lines and circles to create an arc and fillet the objects.

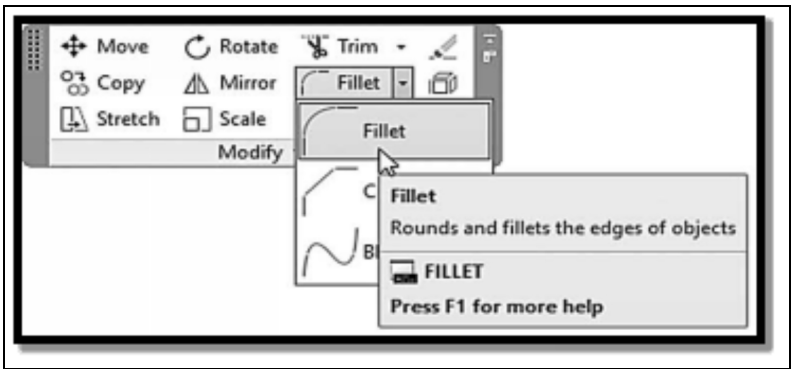



When working with a Radius greater than zero, users have the option to choose between **Trim** or **No trim**. Trim allows for trimming to the original objects, while No trim allows the original objects to remain unchanged. For both scenarios, users will have the ability to visually observe the arc when they hover over the second object, enabling them to verify the accuracy of the value.

**Here’s an example:**




To initiate this command, navigate to the Home tab, find the Modify panel, and click on the **Fillet button**:




**Here are the AutoCAD prompts that will be displayed to the user:**

Current settings: Mode = TRIM, Radius = 0.0000

Select the first object or [Undo/Polyline/Radius/Trim/Multiple]:

Always ensure to review the initial line of this command, as it provides the current value of the Radius. Based on this information, you can determine whether to retain or modify it. To set the new value of the Radius, you can either type "**r**" or right-click and select the Radius option. Upon opening the program, a prompt will be displayed:

Specify fillet radius <0.0000>:

To change the mode to Trim or No Trim, simply type "t" or right-click and select the Trim option. A prompt will be displayed:

Enter Trim mode option [Trim/No trim] <Trim>:

To perform multiple fillets in a single command, you can switch to the Multiple mode. By doing so, you can easily create more than one fillet without the need for separate commands. If an error occurs, simply utilize the Undo option to reverse the most recent action. Press [Enter] to complete the command.

**Two important actions can be performed when using the Fillet command:**

- The Fillet command in AutoCAD can be used to fillet two parallel lines, regardless of the current Radius value.
- When you hold down the [**Shift**] key, Fillet will automatically fillet any two lines with a radius of 0, regardless of the current value set for the Radius.

# How to Chamfer Objects

The Chamfer command is great for creating clean and precise intersections. However, the user needs to prioritize setting the value of Distance (or Distance and Angle) as the initial step. If the value is 0, you can utilize the Chamfer command to extend or trim the lines to the desired intersection point. If the Distance value is greater than zero, Chamfer will generate a sloped edge between the two lines. When working with a non-zero distance, users have the option to choose between Trim, which trims the original objects, or No Trim, which leaves the original objects unchanged. For both scenarios, the user will have the ability to visually observe the chamfer line when hovering over the second object, ensuring that the value is accurate.
**There are two methods you can use to create a sloped edge:**

- Distance and Angle
- Distance (two distances)

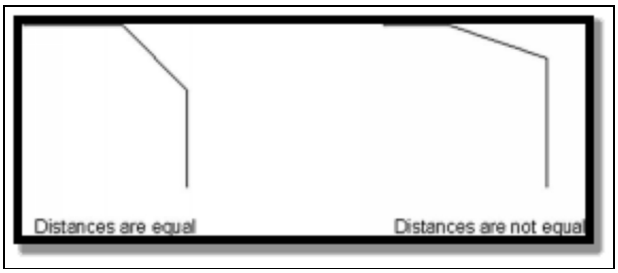
# Chamfering Using the Distance Option

When using the Distance option to chamfer, you will encounter two prompts:

Specify first chamfer distance <0.0000>:

Specify second chamfer distance <0.0000>:

**There are two distinct cases to consider:**

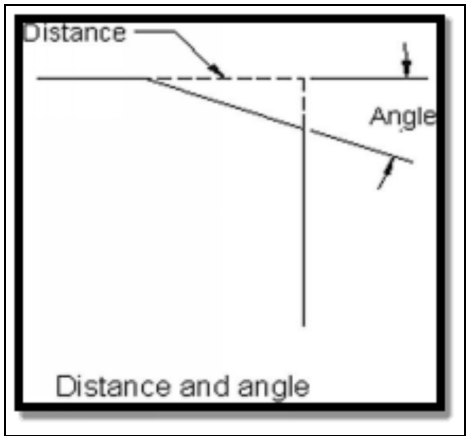



# Chamfering with Distance and Angle

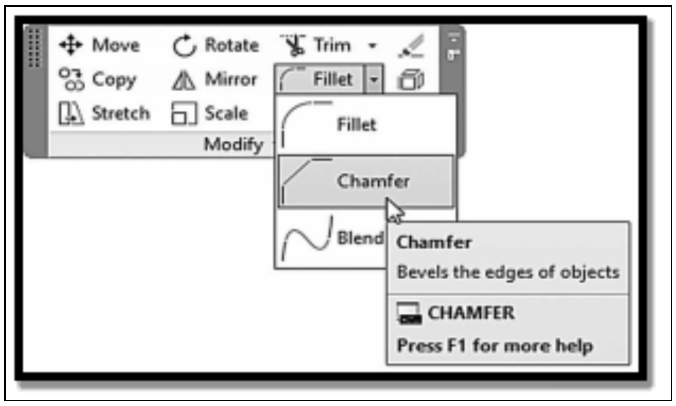
When chamfering using Distance and Angle, you will encounter two prompts:

Specify chamfer length on the first line <0.0000>: Specify chamfer angle from the first line <0>:

**Specify the desired length to be cut from the first object selected, along with the desired angle.**
**Take a look at the illustration below:**




**To initiate this command, navigate to the Home tab and find the Modify panel. From there, simply click on the Chamfer button:**




**The user will be presented with the following prompts:**

(TRIM mode) Current chamfer Dist1 = 0.0000,

Dist2 = 0.0000

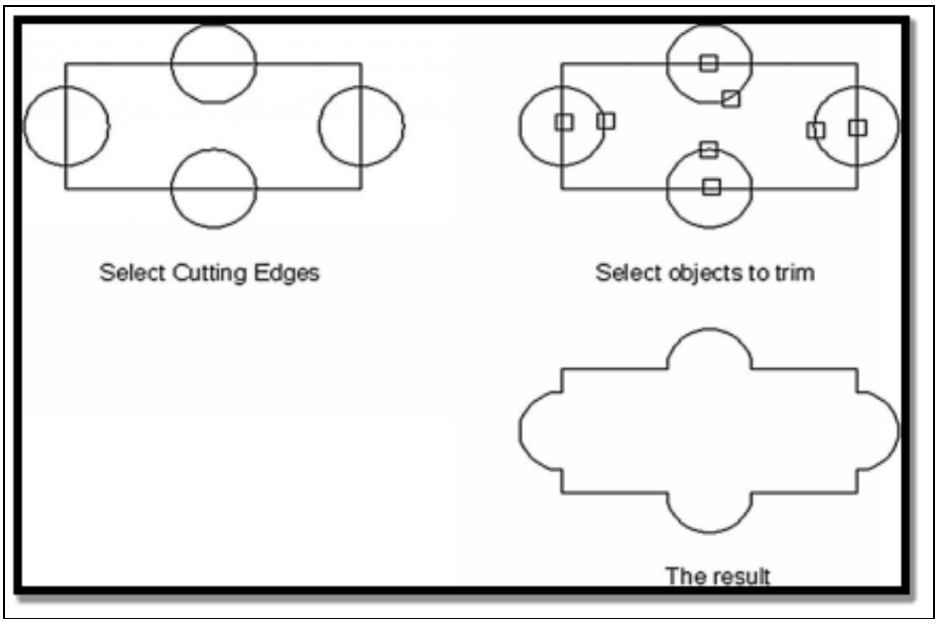
Select first line or [Undo/Polyline/Distance/Angle /Trim/Method/Multiple]:

It is important for users to always review the initial line of this command as it provides the current value of the method being used (either Distance or Distance and Angle), along with the current values. You will need to decide on whether to retain or modify them. Additional features such as Multiple, Trim, and Undo functions in the same manner as those covered in the Fillet command. The Method option allows you to choose the preferred method for chamfering. In a similar fashion to the Fillet command, when you hold down the [**Shift**] key, AutoCAD will chamfer the two lines by extending or trimming them, regardless of the current Distance values. When using Chamfer, it's important to note that the Trim and Untrim functions can have an impact on the Fillet command, and vice versa.

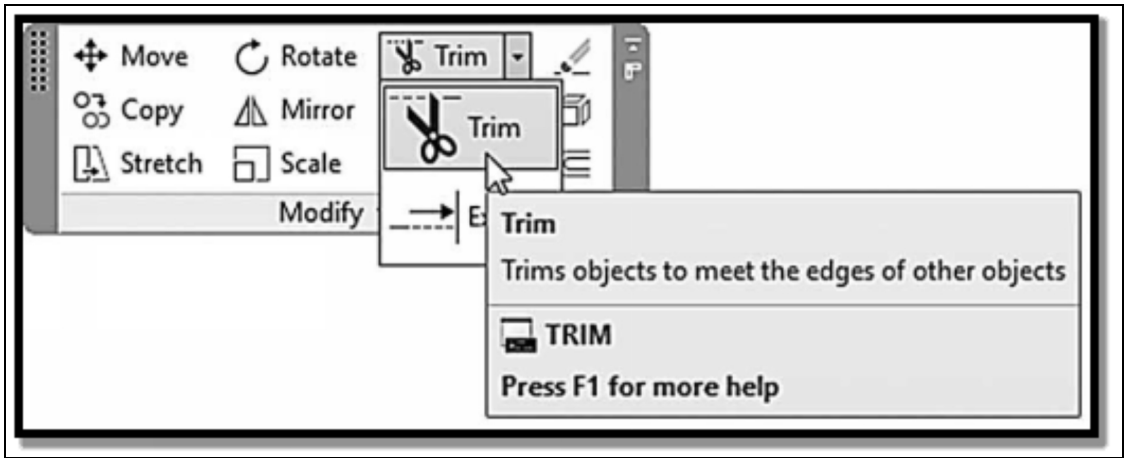
# How to Trim Objects

This command allows you to efficiently remove a portion of an object by utilizing cutting edges. In AutoCAD, the default mode is Quick mode. This mode assumes that all objects in the drawing are cutting edges. To remove a specific part of an object, simply click on that part.

**Here's an example that demonstrates the process of trimming:**




To execute this command, navigate to the Home tab, find the Modify panel, and click on the **Trim button**:




**A prompt will be displayed:**

Current settings: Projection=UCS, Edge=None, Mode=Quick

Select object to trim or shift-select to extend or

[cuTting edges/Crossing/mOde/Project/eRase]:

AutoCAD provides a message displaying the current settings in the first line. Ensure that the current mode is set to Quick. To remove or trim a specific part of an object, simply click on the desired area using the clicking, Fence, or Crossing option. When trimming, it's possible to

encounter an orphan object. To remove it, simply use the eRase option by pressing the letter r.

**Here are the available Mode options:**

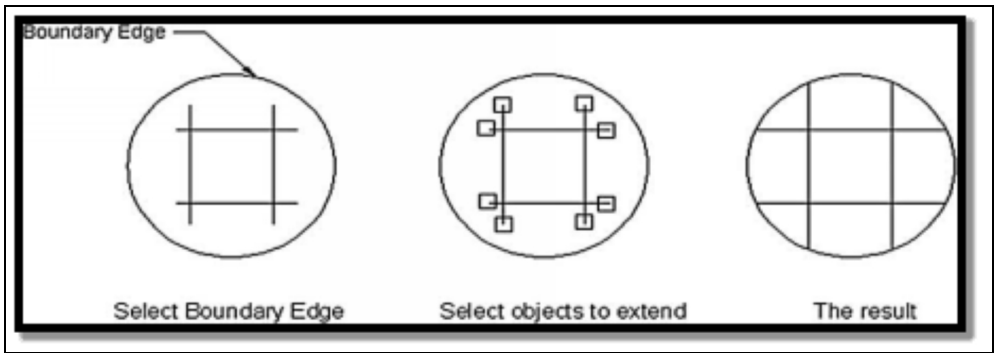
- To switch back to the Standard method, simply type 'O' for mode. This will prompt the following message:

Enter a trim mode option [Quick/Standard] <Quick>: S

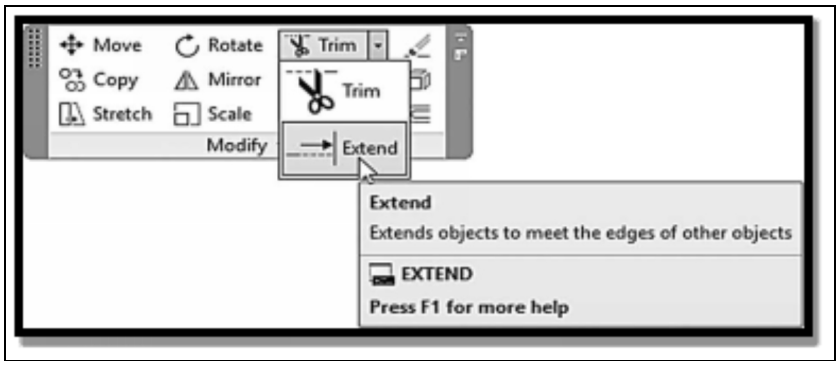
When using the Trim command, a prompt will appear:
Current settings: Projection=UCS, Edge=None, Mode=Standard
Select cutting edges ...
Select objects or [mOde] <select all>:
When working in AutoCAD, it is important to specify the Cutting Edges before proceeding. Alternatively, you can simply press [Enter] to select all the objects that will serve as cutting edges. The default mode will be Standard until you choose to switch it back to Quick. When you enter T in both modes, you have the flexibility to specify your cutting edges according to your preferences.

# How to Extend Objects

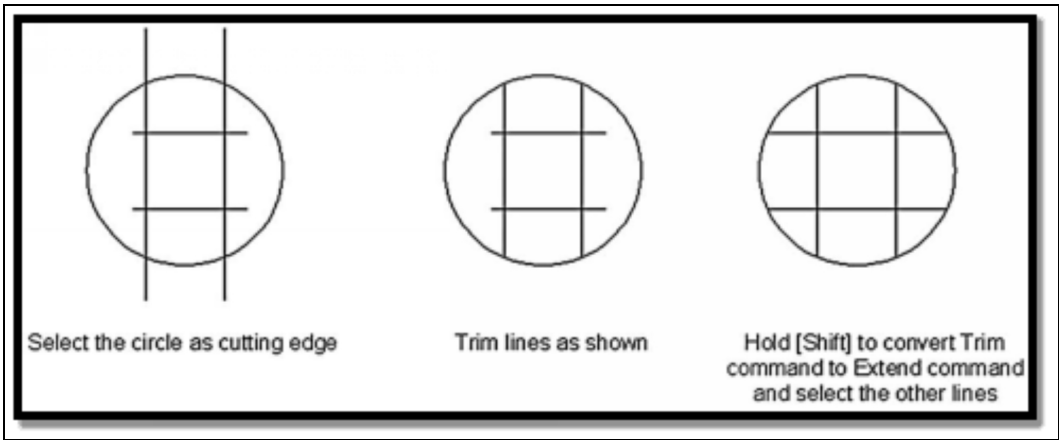
To extend an object to a boundary edge, you can use this command. In AutoCAD, the default mode is Quick mode. This mode assumes that all objects in the drawing are boundary edges. To extend an object, simply click on the part of the object that you want to extend. **Take a look at the accompanying diagram:**




To execute this command, navigate to the Home tab, find the **Modify panel**, and click on the **Extend button**:

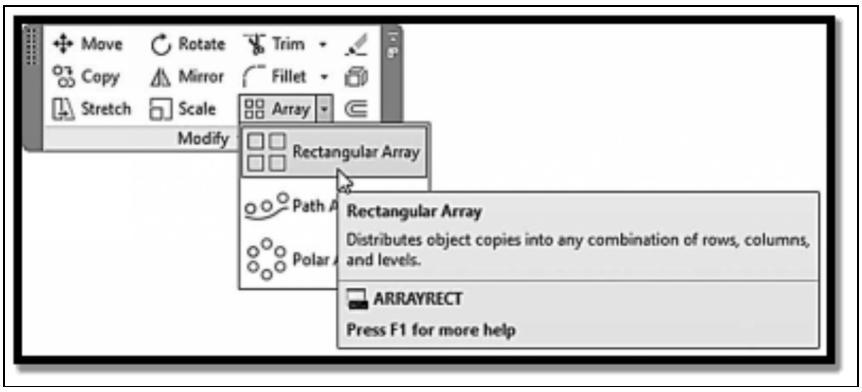



The Trim command provides a comprehensive overview of all available options and prompts. One useful feature of both the Trim and Extend commands is the ability to use them simultaneously. **Take a look at the following example:**




# Arraying Objects – Rectangular Array

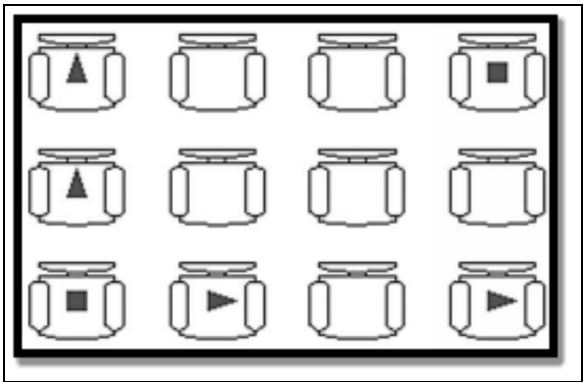
This command allows you to easily generate replicates in a matrix format by specifying the number of rows and columns. The final shape will become a single object that is editable. To execute this command, navigate to the Home tab, locate the Modify panel, and click on the **Rectangular Array button**:



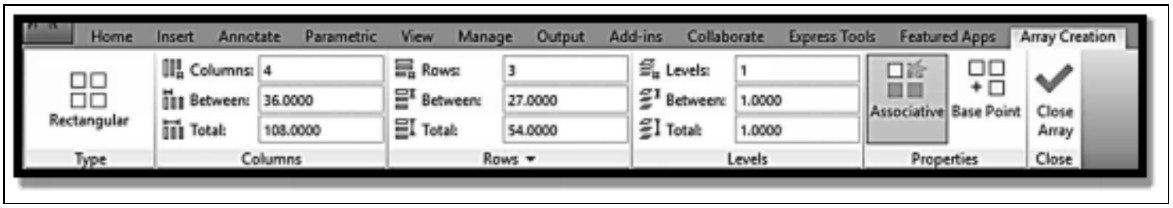

**Here is the prompt that will be displayed:**

Select objects: 1 found

After selecting the desired objects, simply press [**Enter**] and AutoCAD will promptly display a grid with three rows and four columns, complete with grips for easy manipulation.



Afterward, you'll notice a context tab called **Array Creation**:




**When you initiate the command, a series of prompts will be displayed in the command window:**

Type = Rectangular Associative = Yes

Select grip to edit array or [ASsociative/Basepoint/COUnt/Spacing/COLumns/Rows/Levels/eXit]

<eXit>:

**Please select the appropriate grip in AutoCAD to make changes to the array. To achieve the desired result, the user is encouraged to utilize two tools:**

- Use the Array Creation context tab to perform the tasks mentioned above.
- Use grips to easily adjust the number of rows, columns, and distances between columns and rows, as well as the direction of arraying, whether it be downward or upward, right or left.

# How to Use the Array Creation Context Tab

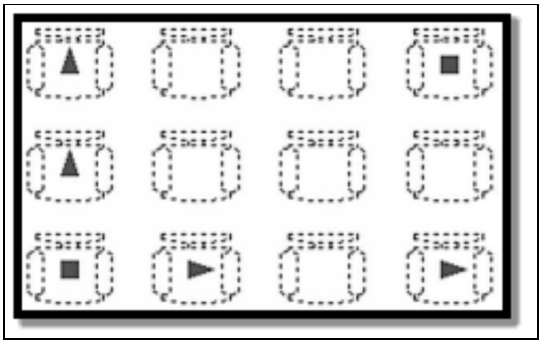
Using the Array Creation context tab becomes much more convenient when you have a solid understanding of the numbers and distances involved. **First, let's begin by utilizing this method:**

- Use the Column panel to input any two out of three pieces of information: Columns (number of columns), Between (distance between columns), and Total (total distance the columns will occupy). Consistency is key when it comes to measuring distances in AutoCAD. It's important to always use the same reference point, whether it's from left to left or from center to center. Additionally, it is important to take into account the direction of the arraying. Positive distances indicate upward movement, while negative distances indicate downward movement.
- Use the Rows panel to enter two out of three pieces of information: Rows (number of rows), Between (distance between rows), and Total (total distance the rows will occupy). Consistency is key when it comes to measuring distances in AutoCAD. It is important to always use the same reference point, whether it's from top to top or from center to center. This will ensure accuracy and precision in your designs. Additionally, it is important to take into account the direction of the arraying. Positive distances indicate moving to the right, while negative distances indicate moving to the left.

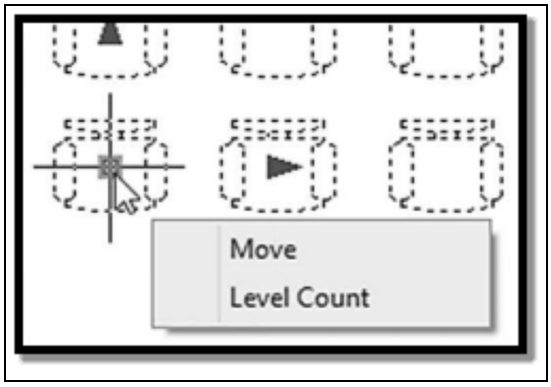
- Levels are not necessary for 2D work as they are primarily used in 3D modeling.
- When objects are created using an array, they are considered as a single entity that contains all the information used to construct them. This is known as being associative.
- In AutoCAD, you have the flexibility to choose a different object as the base point for the array, instead of the default first object.
- To end the command, simply click on the **Close Array button**.

# How to Edit a Rectangular Array Using Grips

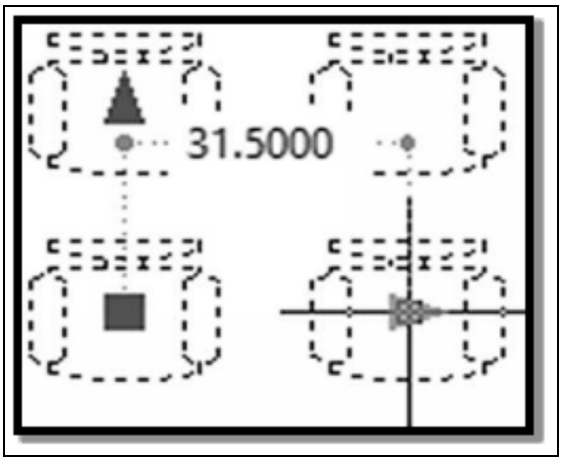
**Once a rectangular array is inserted, simply select any object to make edits. The objects will display grips as shown below:**



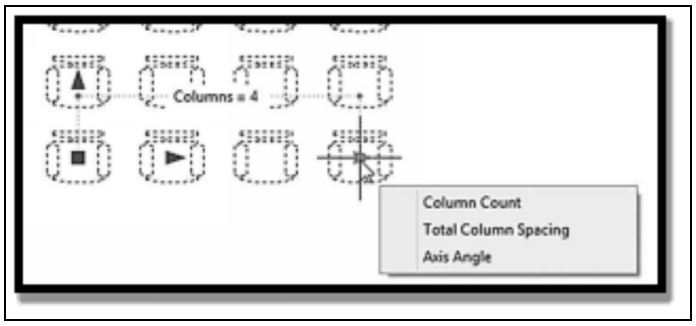
**Every single one of these commands serves a specific purpose:**



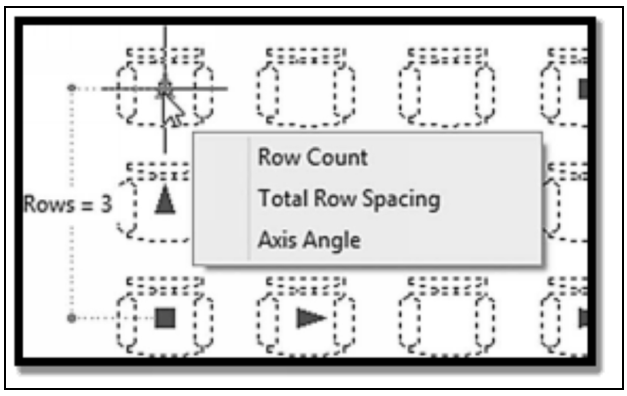

This grip allows for the manipulation of the entire array object or the adjustment of the level count in 3D designs. Utilize the [Ctrl] key to navigate through these various options.



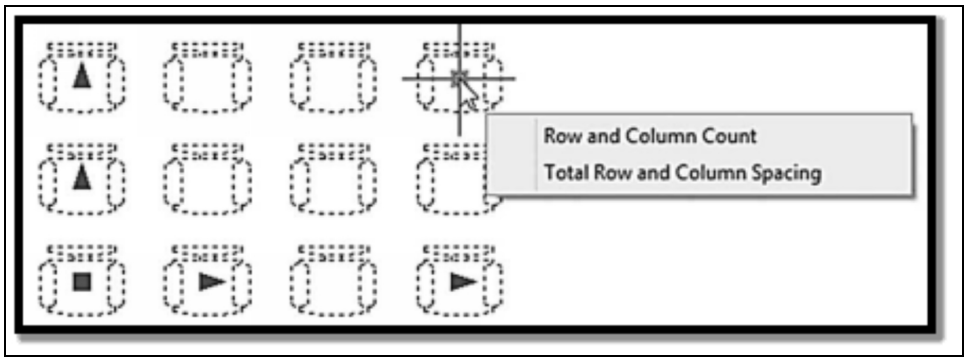

This grip is specifically designed for adjusting the column spacing.




This grip allows you to modify the Column count, Total Column Spacing, or Axis Angle (for angles other than horizontal and vertical). Use the [**Ctrl**] key to navigate through these options.



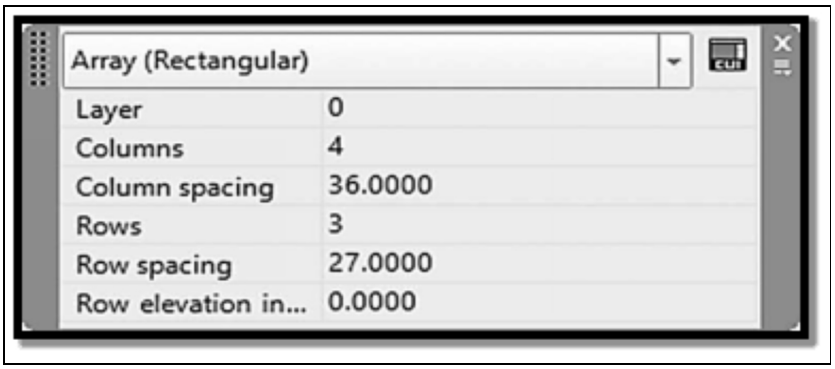

This grip is used to modify the Row Count, Row Spacing, or Axis Angle. Use the [**Ctrl**] key to navigate through these various options.




This grip can be used to modify the Row and Column Count or adjust the Total Row and Column Spacing. Use the [**Ctrl**] key to navigate through these various options.

# How to Edit a Rectangular Array Using Quick Properties

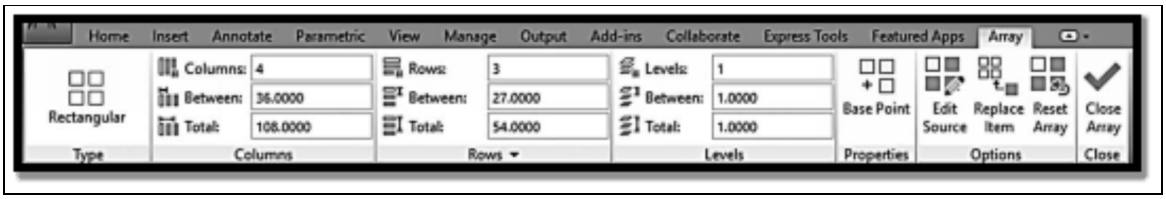
**After clicking on a rectangular array, Quick Properties will be displayed as shown below:**




It is possible to modify the number of columns, adjust the spacing between columns, change the number of rows, and adjust the spacing between rows.

# How to Edit a Rectangular Array Using the Context Tab

**After clicking on a rectangular array, you will notice the appearance of a context tab called Array, as shown below:**

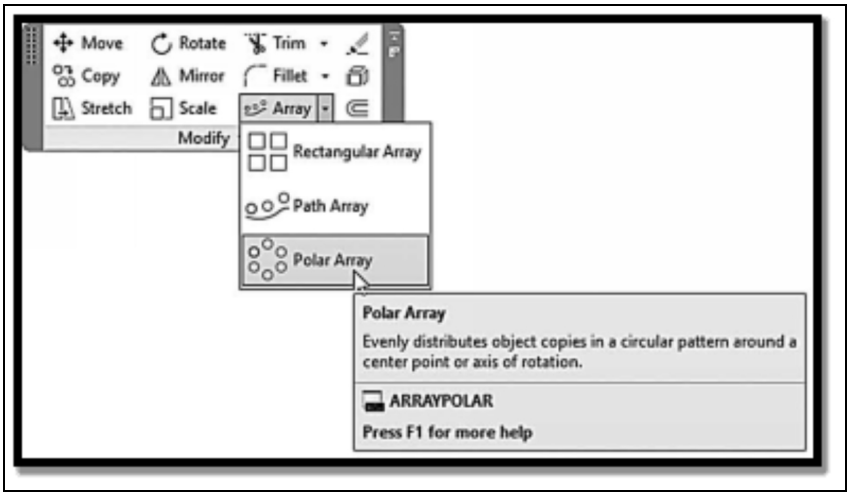



**The tab is very similar to the Array Creation context tab, with the only difference being the Options panel that contains three buttons:**

- The **Edit Source command** allows for modifications to be made to one of the objects in the array, which will then be applied to all other objects in the array.
- The **Replace Item command** allows for the replacement of one or more arrayed shapes with a different shape.
- The **Reset Array command** can be used to undo the changes made by the Replace Items command.

# Arranging Objects – Polar Array

Use this command to create circular duplicates of objects. To execute this command, navigate to the Home tab, find the **Modify panel**, and click on the **Polar Array button**.



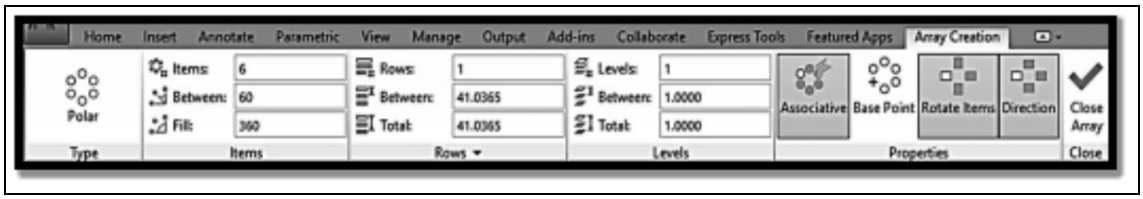

**Here are the prompts that will be displayed:**

Select objects:

Type = Polar Associative = Yes

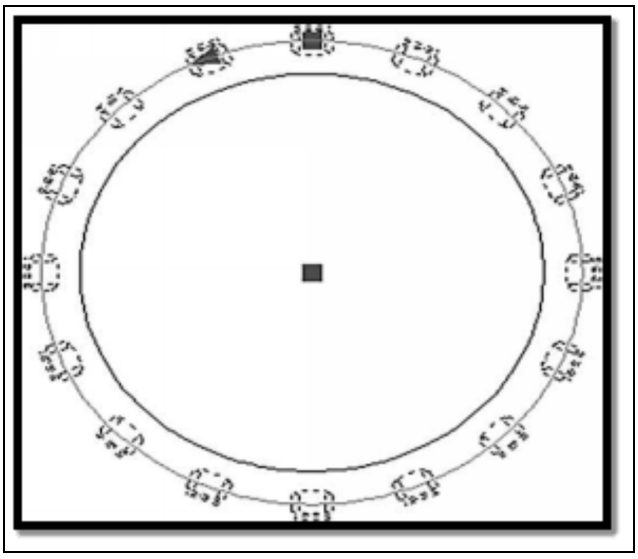
Specify the center point of array or [Base point/Axis of rotation]:

Select the objects you want and then hit [**Enter**] to proceed. In the second prompt, a message will be displayed indicating that the array type is polar and the associativity is turned on. For the third line, you'll need to indicate the center point of the array. Once you've specified the center point, a polar array with six objects that fill 360 degrees will be generated. Additionally, a new context tab called Array Creation will be visible, displayed as follows:

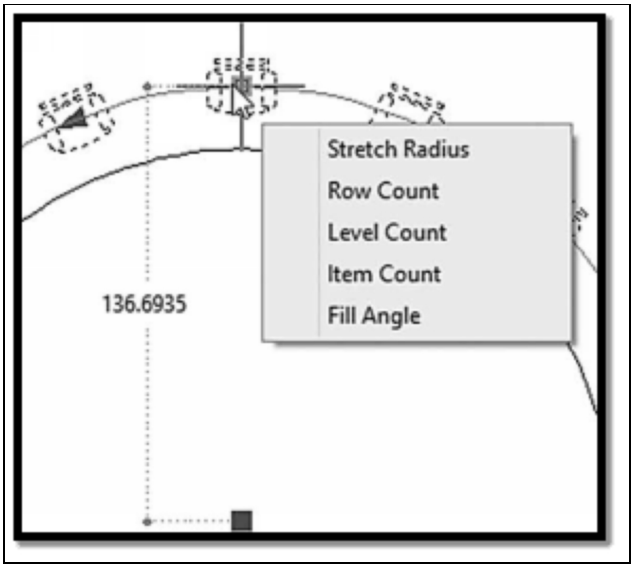



By using the context tab, users can define the quantity of items, the angle between them, and the Angle to fill. Feel free to indicate the desired number of rows and the spacing between them. When working in AutoCAD, you can utilize the Properties panel to easily input the necessary details. You have the option to indicate whether the array will be associative or not. Additionally, you can specify a new base point for the object to be arrayed, and decide whether to rotate items as you copy them. Lastly, you can specify the direction of arraying, either clockwise (CW) or counterclockwise (CCW). These features allow for precise control over your arrays in AutoCAD. With AutoCAD, you have the option to choose polar arraying for more advanced editing capabilities.

**After selecting it, you will see an image that closely resembles the one shown below:**

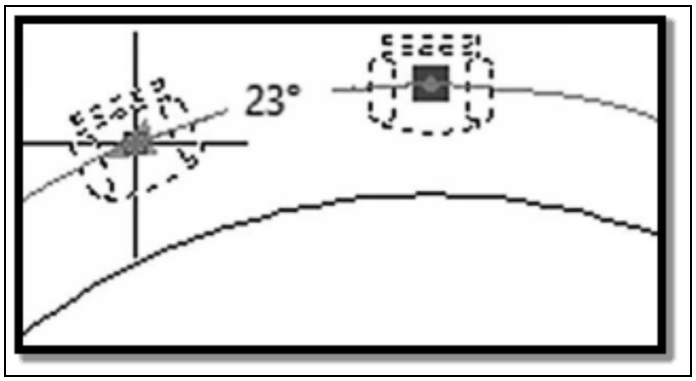


**The initial grip will display the following:**

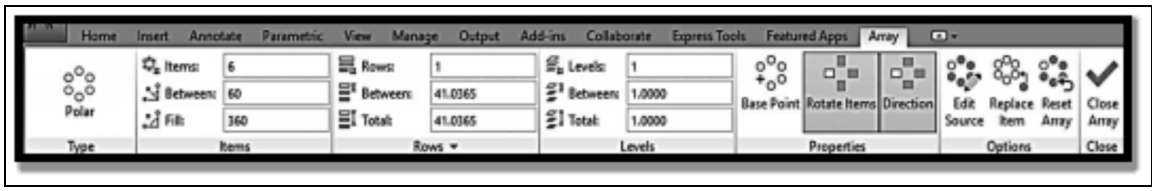



Within the interface, you'll find the radius value displayed alongside a convenient menu. This menu grants you the ability to effortlessly adjust the radius, modify the row count, alter the level count (applicable to 3D designs), manipulate the item count, and even tweak the fill angle to your liking.

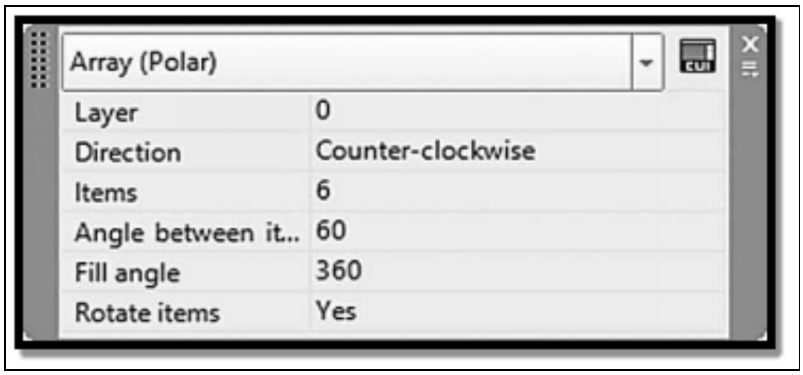
**Here's a simple example of how to modify the row count:** The second arrow grip will display the precise angle between the first and second items.




**Additionally, a context tab called Array will appear:**




**This is the same as what we covered in the rectangular array. Additionally, upon selecting the polar array command, the Quick Properties panel will be displayed, showcasing the following options:**

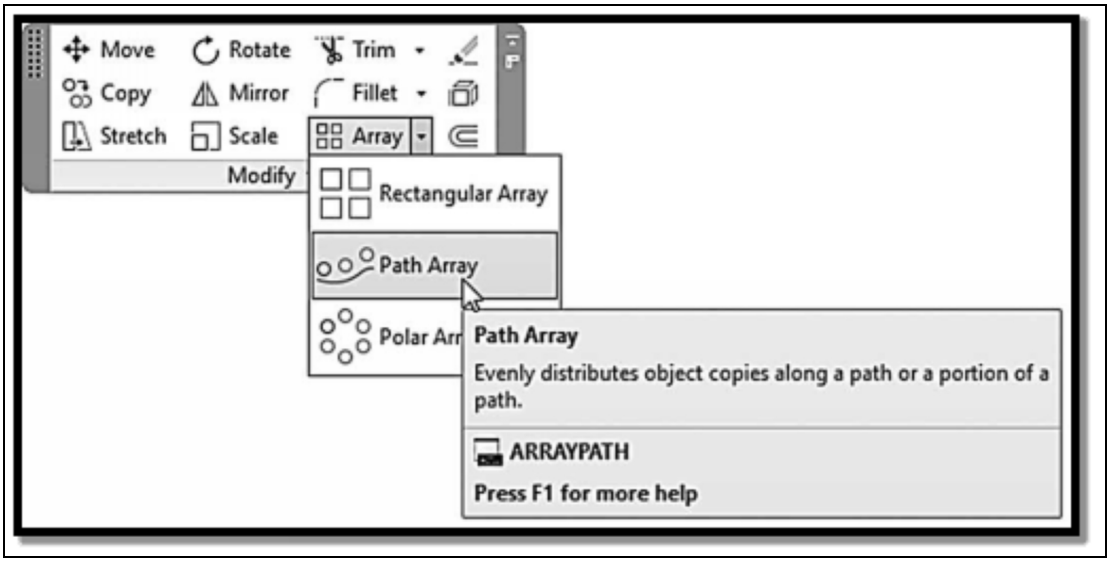



Within Quick Properties, users can modify the array's direction (clockwise or counterclockwise), adjust the quantity and angle between items, alter the overall fill angle, and decide whether or not to rotate items during the copying process.

# Arranging Objects – Path Array

The array command in AutoCAD allows you to create multiple copies of an object, such as a polyline, spline, arc, and more. To execute this command,

navigate to the Home tab, find the Modify panel, and click on the **Path Array button**.




**Here are the prompts that will be displayed:**

Select objects:

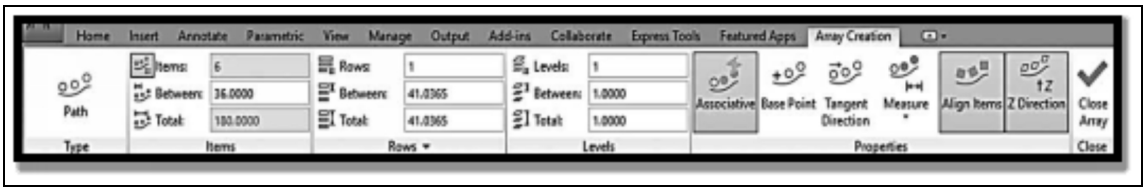
Type = Path Associative = Yes

Select path curve:

When prompted, you will be asked to select the objects you want. Once you have made your selection, simply press [**Enter**]. The second prompt will display a message indicating that the array type is path and the associativity is turned on. After the user selects the path curve, AutoCAD will display ten objects arranged along the path. A prompt will appear:

Select grip to edit array or [ASsociative/Method/Basepoint/Tangent direction/Items/Rows/Levels/Align items/Z direction/eXit]<eXit>:

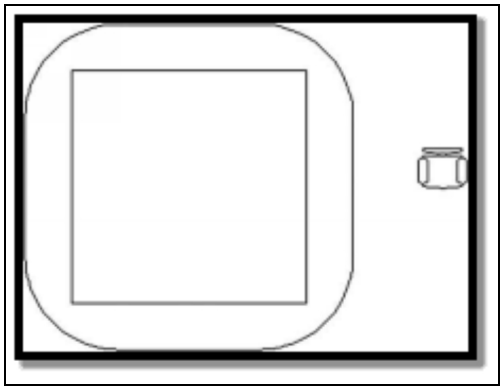
**Next, you'll notice the Array Creation context tab, which appears like this:**



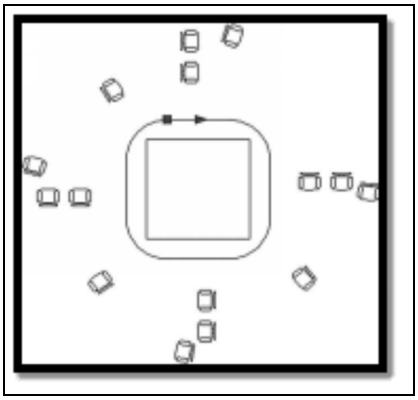
**When considering the process of arranging an object along a path, there are three key factors to keep in mind:**

- Measure or Divide
- Basepoint
- Aligning objects with the path

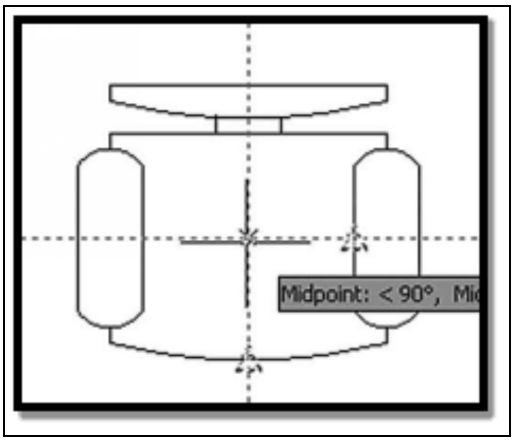
**Now, let's examine how AutoCAD will handle the given scenario:**



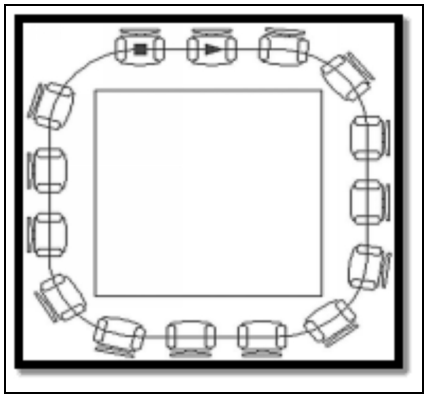
**For the array operation, we'll choose the chair as the object and the outer polyline as the path. Observe how AutoCAD will handle this situation:**



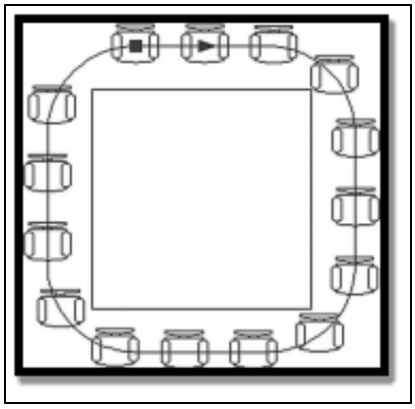
**Next, navigate to the context tab and locate the Base Point option. This will allow you to designate a new base point, which in this case will be the center of the chair.**




**Here is the outcome once the new base point has been specified:**

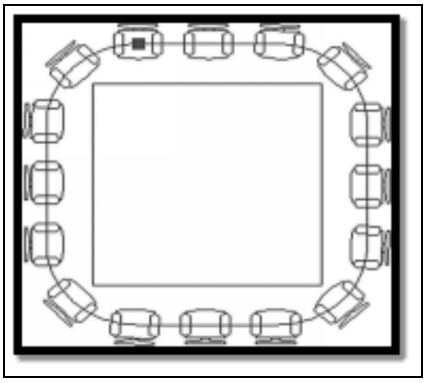


**Naturally, the Align button is enabled by default. If you choose to disable it, this is the result you will obtain:**

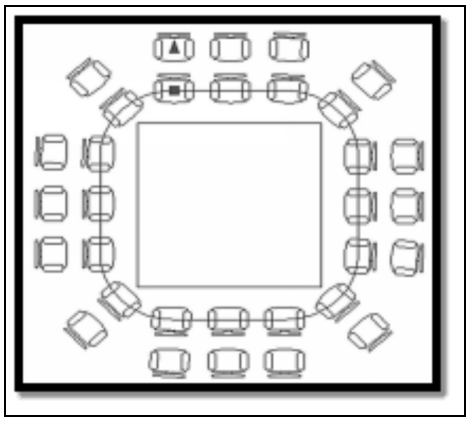


When you choose the Divide option, AutoCAD will evenly divide the path for all the objects. However, if you opt for Measure, you will need to indicate the distance between each object. The first result was generated

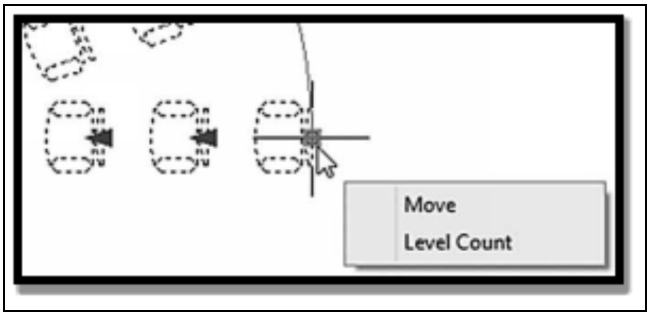
using the Measure tool, while the second one was obtained using the Divide function.



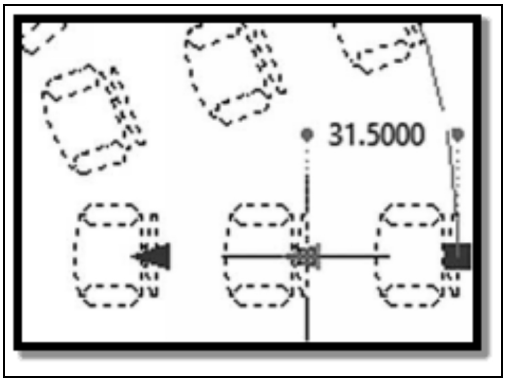
Once the three factors have been specified, you can easily choose the number of items, the distance between them (which may be adjusted if using the Divide command), and the total distance. In addition, you have the option to include additional rows to accommodate the following:



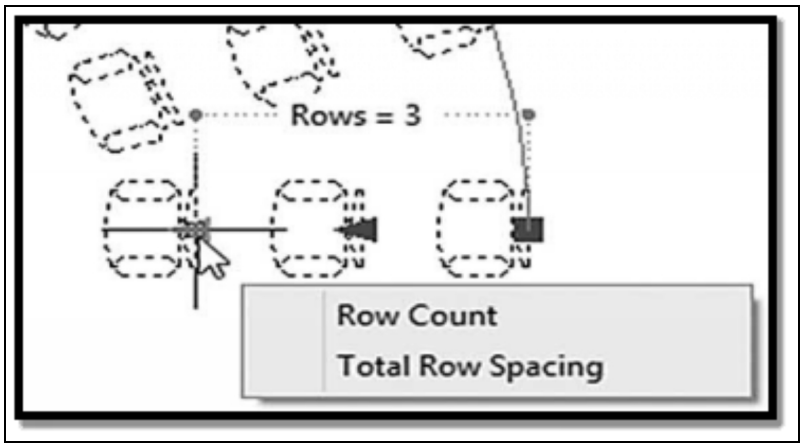
Once the path array is created, you have the option to select it for editing. When using the Divide command, you will notice three grips on the screen. However, in the Measure command, there will be an additional arrow displayed. Here is the first one:



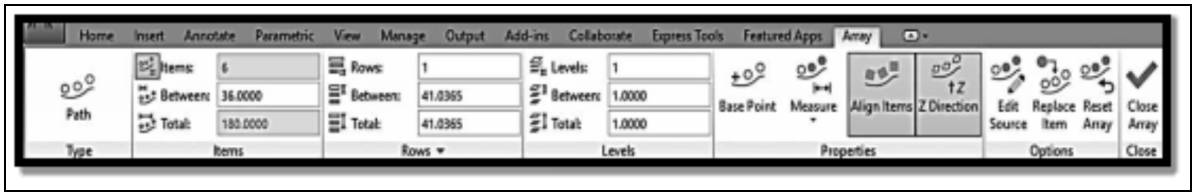
You can easily manage both the movement and level count with this.
**Here is the second option:**



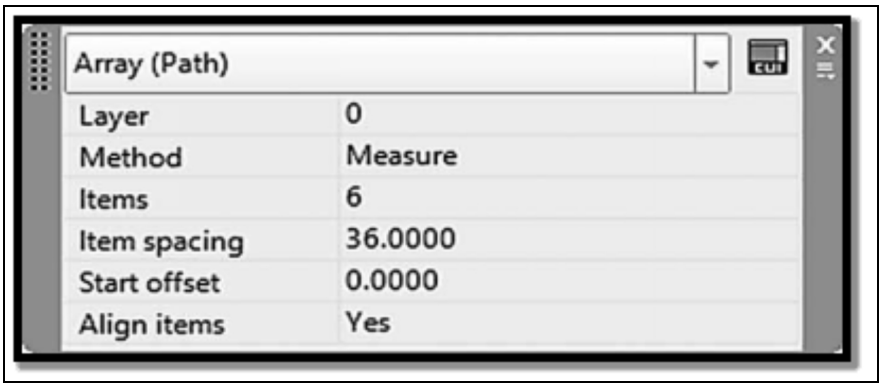

**This grip will define the spacing between rows. Here is the third option:**




You can use this to manage the number of rows and the spacing between them. **Additionally, a new context tab called Array will be displayed, with the following appearance:**



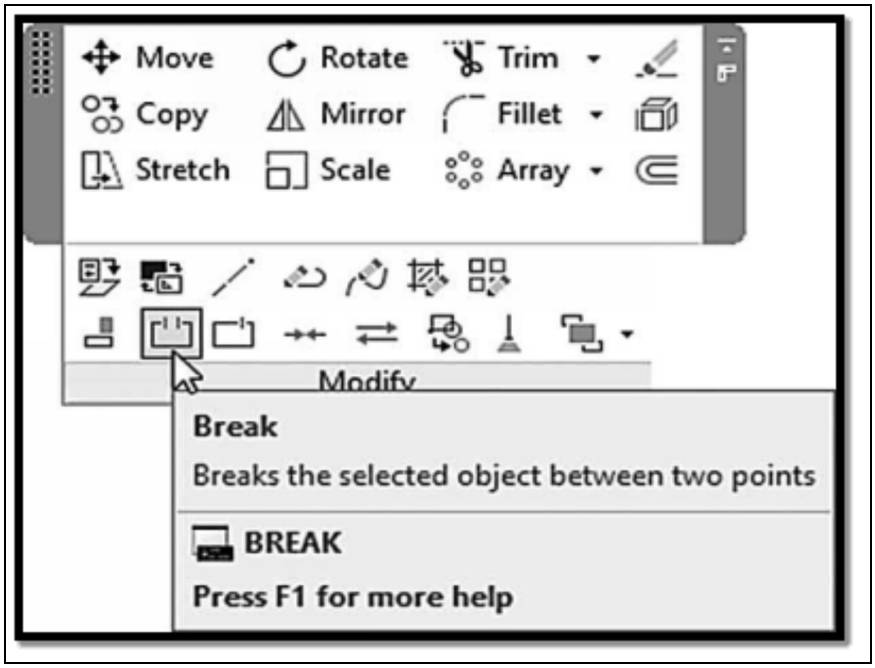

This is the same as what we covered in the rectangular array. **In addition, once a path array is selected, the Quick Properties panel will be displayed:**




Within Quick Properties, you can modify various settings. These include the Method (Divide or Measure), the number of items, the spacing between items, the Start Offset (which determines where the array begins along the path), and the option to align items during the array process.

# How to Use Break Command

With this command, you can effortlessly split any object into two separate entities by eliminating the section between two designated points. To execute this command, navigate to the Home tab, find the Modify panel, and click on the **Break button**:




**After launching the program, a series of prompts will be displayed for your attention:**

Select object:

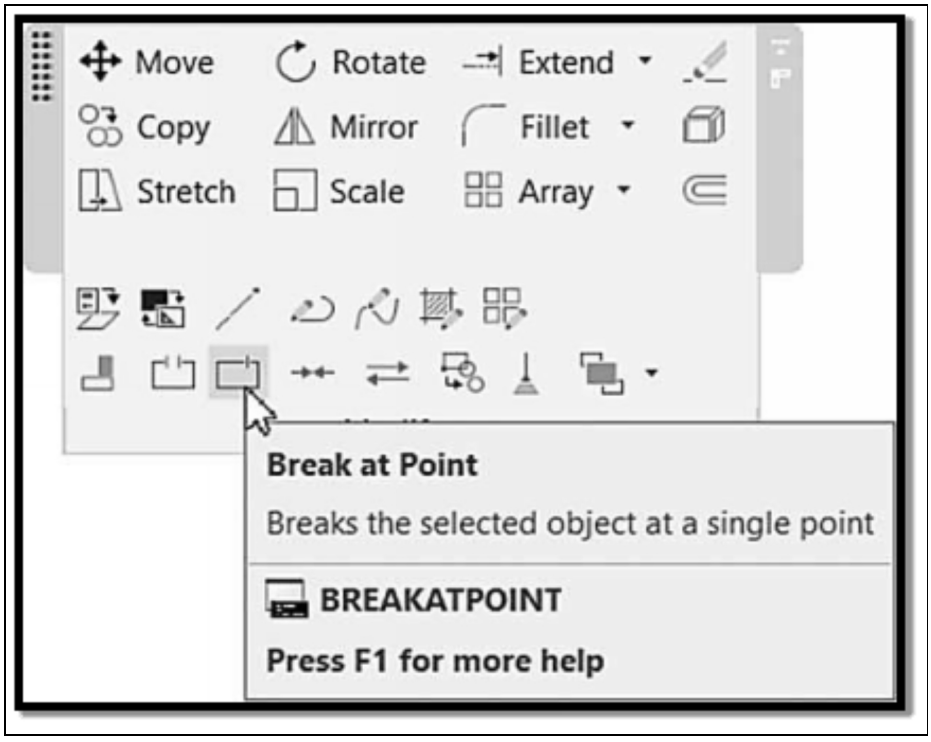
Specify second break point or [First point]:

When selecting an object, remember to specify the first point and then proceed to select the second point. Executing this command will conclude the process. However, if you approach the selection process as simply selecting without specifying the initial point, you can type F to view the subsequent two prompts:

Specify first break point:

Specify second breakpoint:

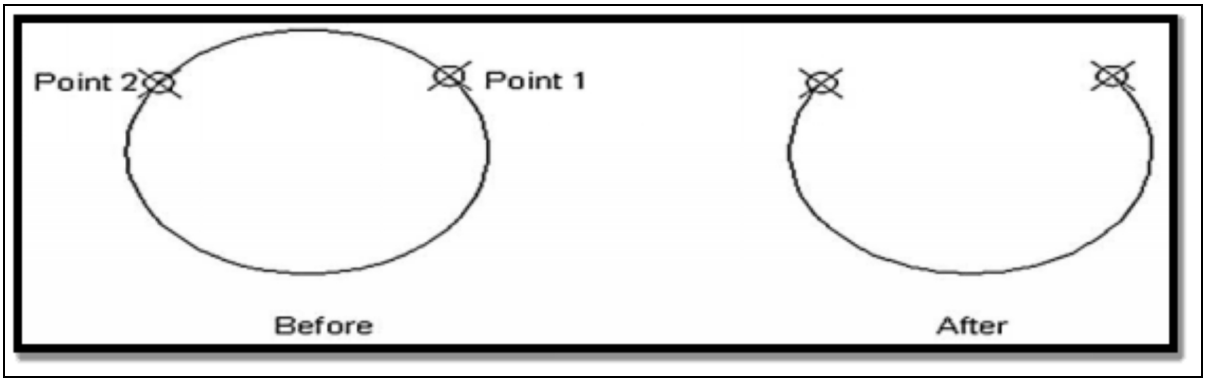
With the help of these two prompts, you can specify two points on the object, and the Break command will terminate.
AutoCAD provides an alternative command that uses a distinct method for breaking at the identical point. To execute this command, navigate to the **Home tab**, find the Modify panel, and click on the **Break at Point button**.




**After opening the program, a series of prompts will be displayed for your attention:**

Select object: Specify breakpoint:

To break an object in AutoCAD, simply select it and then indicate the point where you want it to be divided.
Take note of the following:
When breaking a circle, it is important to specify the two points in a counterclockwise direction. **Take a look at this example:**




*OceanofPDF.com*

# CHAPTER FIVE
# CIRCLES AND DRAWING AIDS

Here, you will learn how to take control of your drawing environment by adjusting the grid and snap spacing, as well as the units used to display coordinates. Expanding your collection of objects is easily accomplished by using the CIRCLE command for drawing circles and the RECTANG command for creating rectangles. As you delve deeper into the world of AutoCAD, you will become well-versed in the various techniques of object selection. This knowledge will prove invaluable as you master the intricacies of editing procedures, particularly when using the ERASE command.

## How to Change the Grid Setting

**Here is a step-by-step guide on how to change the grid spacing using the command line prompt:**

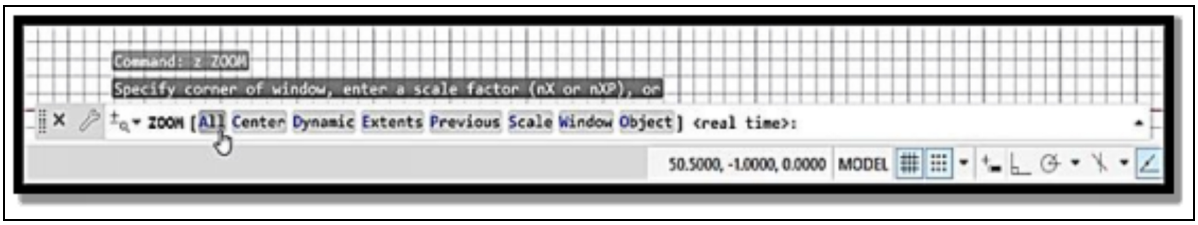
- Press the **Enter** key after typing "**grid**".
- Input a new value.
- When starting a new drawing using the AutoCAD template, the grid and snap settings are automatically configured with a spacing of 0.5000 units. Drawings can be completed without making any changes to the grid and snap settings, which are already set to their default values. Typically, it is advisable to modify this value to align with the requirements of your specific project. For a mapping project, you may find it useful to have a 10-mile snap, while for a printed circuit diagram, a 0.010" snap would be appropriate. You have the option to align the grid with the snap setting or adjust it separately to your preference.
- To create a new drawing, access the Start tab and click the down arrow on the New drop-down list. From there, choose **acad.dwt**. If the acad.dwt file is not appearing on the drop-down list, you can select it by clicking on Browse templates and then choosing the acad.dwt file before clicking **Open**. When you click the **New**

**button** without selecting the drop-down list, a new drawing is created using the most recently used drawing template.

- Enable Snap Mode and Grid Display by using the buttons located on the status bar.
- To initiate the ZOOM command, simply press the letter "z" followed by the Enter key.

# Selecting Options from the Command Line

Here, you will learn the process of selecting an option from the command line. Take note of the blue highlighting on the uppercase letters in each option of this prompt from the ZOOM command. To execute the All option, simply type the letter A and press <**Enter**>. Alternatively, you can click on the word **All** in the command line.




**Choosing options from the command line**

- Click on the word "**All**" in the command line. The grid has been properly aligned and the zoom level has been adjusted to fit the entire drawing. To access the options displayed on the command line, simply select the desired option.
- Press the **Enter** key after typing "**grid**". When using AutoCAD, the command-line prompt will be displayed in a specific format. It will show options that are separated by spaces. Specify the desired grid spacing for the X-axis or choose from the available options: ON, OFF, Snap, Major, or Adaptive. Set the limits to 0.5000:

When dynamic input is enabled, a portion of the prompt will be displayed alongside the crosshairs. Feel free to disregard the options for the time being. The value of 0.5000 represents the current setting.

- If you want to see the prompt's response, simply enter the number 1 and observe the outcome. The drawing area will display a grid with a unit size of 1.
- Experiment with different grid spacing values. Consider using the values 2, 0.25, and 0.125. It's important to note that the last command, GRID, can be repeated simply by pressing <**Enter**> or the spacebar.
- Before proceeding to the subsequent section, reset the grid to 0.5000.

# How to Change the Snap Setting

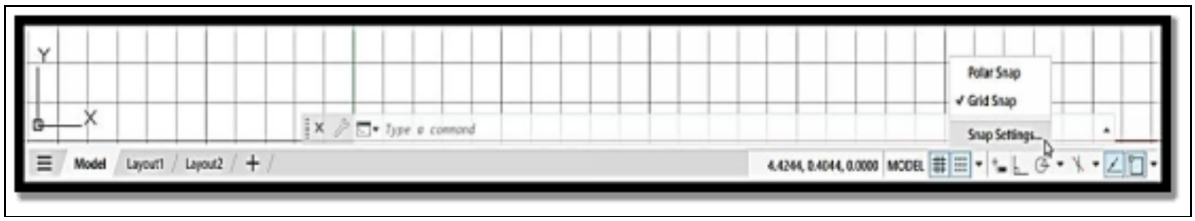
There can be some confusion between grid and snap due to their similarities. The grid serves as a visual reference in AutoCAD. Specifying points is not affected by this. The snap feature in AutoCAD may not be visible, but it plays a crucial role in accurately specifying points. It is important to note that the values of grid and snap may not always be the same. **Here is a step-by-step guide on how to modify the snap setting in the Drafting Settings dialog box:**

- To access the Snap Settings, simply right-click on the **Snap Mode button** located on the status bar.
- Go to the Snap and Grid tab and input a new value for snapping.
- Click on the OK button to close the dialog box.

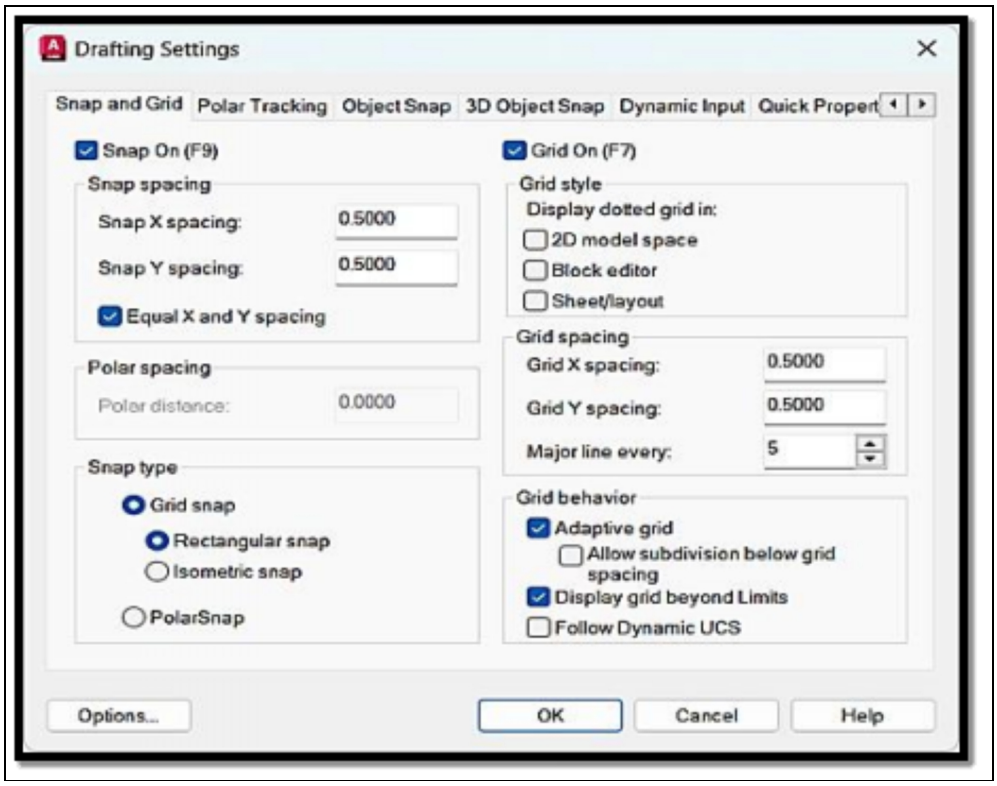
# Using the Drafting Settings Dialog Box

To modify the snap settings, use the **SNAP command** at the command line, similar to how you adjusted the **GRID command** in the previous section. You can also make changes to both options in the Drafting Settings dialog box, just like you're doing here.

- To access the Snap Settings, simply right-click on the Snap Mode button located on the status bar. This option can be found in the shortcut menu.




The dialog box can be displayed using the SETTINGS command, which also has the alias DS. Take a moment to observe the dialog box. It includes several standard features such as tabs, checkboxes, panels, and edit boxes. After opening the dialog box from the Snap Settings item on the shortcut menu, you will find that the Snap and Grid tab is already selected. If you haven't done so already, click it now.




- To ensure the Snap and Grid tab is current, simply click on the tab.

Located below the tabs are checkboxes that are labeled Snap On (F9) and Grid On (F7). To enable or disable snap and grid, simply select the corresponding checkbox. Ensure that the checkboxes in this dialog box indicate that both snap and grid are enabled. You can easily set both the x

and y spacing in the dialog box. If you ever need to have a grid or snap matrix with varying horizontal and vertical increments, AutoCAD can accommodate that. Dialog box commands, like other commands, can be easily repeated by simply pressing the spacebar or <**Enter**>. The snap and grid settings can be found in the edit boxes labeled Snap X spacing, Snap Y spacing, Grid X spacing, and Grid Y spacing. To highlight the entire text in the edit box, simply double-click. Alternatively, you can perform partial editing by clicking once anywhere inside the box.

**To modify the snap setting, follow these steps:**

- Locate the edit box labeled Snap X spacing and double-click inside it.
- Ensure that the entire value of 0.5000 in the Snap X spacing box is selected.
- Enter "**Type 1**" and press the **Enter key**. Pressing the Enter key at this point will have the same effect as clicking OK in the dialog box. Exiting the dialog box will return you to the drawing area.
- The snap value has been adjusted to 1, while the grid remains set at 0.5. The snap setting is set to a larger value compared to the grid setting.
- To access the Line tool, you can either press L or navigate to the Draw panel on the Home tab of the ribbon and click on it.
- Initiate the LINE command to begin. Make sure to activate a drawing command before using snap to specify points.
- Manipulate the cursor within the drawing area. Only half of the grid line intersections are accessible. This arrangement lacks practicality and usefulness. Consider experimenting with different configurations.
- To bring up the dialog box once more, simply right-click on the **Snap Mode button** located on the status bar. From the shortcut menu that appears, select Snap Settings.
- Adjust the Snap X spacing value to 0.25 and then proceed to close the dialog box by clicking on OK.
- Ensure that the cursor is moved at a controlled pace while closely monitoring the coordinate display. This arrangement is optimized for maximum efficiency. By setting the grid larger

than the snap, you can effortlessly specify precise points without the grid becoming overly cluttered.

- Adjust the Snap X spacing value to 0.05. Manipulate the cursor and observe the coordinate display.
- Take note of how the snap setting affects the displayed coordinates. What is the minimum size of a snap that AutoCAD can accept?
- Consider using a value of 0.005. Manipulate the cursor and carefully observe the coordinate display.
- Consider using a value of 0.0005. Consider experimenting with a value of 0.0001, although keep in mind that this may result in a similar effect as disabling the snap feature since the coordinate display already shows four decimal places. Unlike the grid, the snap feature in AutoCAD allows you to customize the value to your preference, without being restricted by the size and resolution of your screen. When attempting a grid setting that is too small, AutoCAD will automatically switch to a larger grid.
- To exit the LINE command, simply press the <**Esc**> key.
- Before completing this exercise, make sure to set the snap back to 0.25 and keep the grid at 0.5.

To maintain consistent snap and grid settings, you can either set the grid to 0 in the Drafting Settings dialog box or use the **GRID command** and select the **Snap option** by typing S. When you reset the snap, the grid will automatically adjust to match the new snap settings and will continue to update accordingly. To unlock the grid, simply assign it a new value.

# How to Adjust Units

In a drawing, distances are always unitless. This means that 1 drawing unit can represent various measurements such as millimeters, inches, meters, feet, kilometers, or even miles. Architectural drawings often include measurements in meters, millimeters, feet, or inches, while mechanical drawings typically use decimal formats for measurements. With the UNITS command, you have complete control over the format of length and angular values for display and input. After initiating the UNITS command, the Drawing Units dialog box will be presented. **Here is a step-by-step guide on how to change units in AutoCAD:**

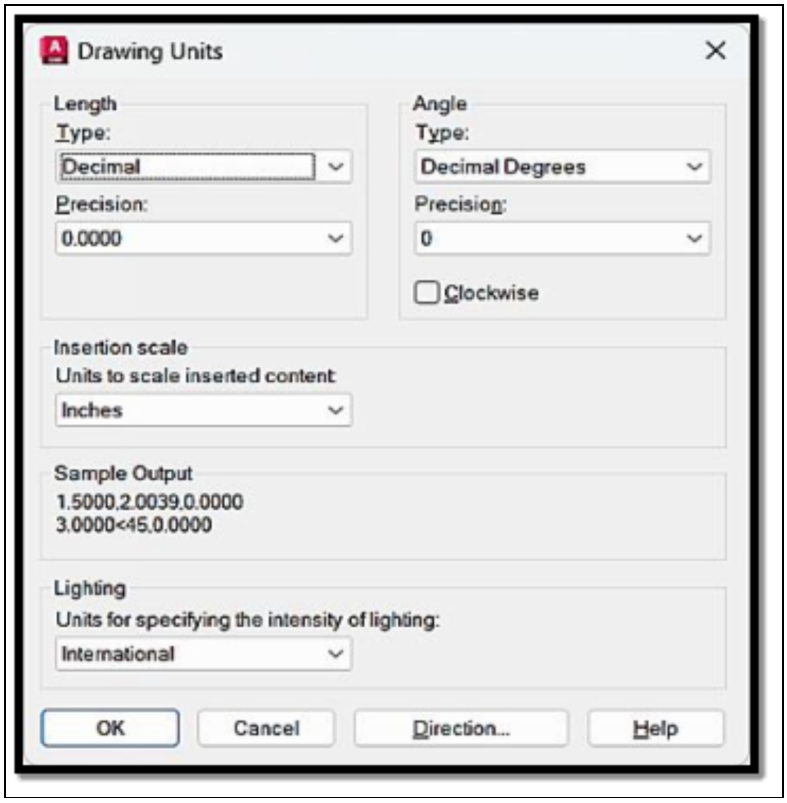
- Enter the "**Type Units**" command. Provide your answers to the prompts.

# The Drawing Units Dialog Box

**The Drawing Units dialog box uses drop-down lists, which is a commonly used feature in dialog boxes.**

- To access the Units feature, you can either press Enter or navigate to **Application > Drawing Utilities > Unit**s. The Application menu can be found in the upper-left corner of the application.

Drop-down lists display the current setting alongside an arrow that allows you to access a list of alternative options. Ensure that the dialog box displays the current Length Type in your drawing as decimal units with a precision of 0.0000 places and the Angle Type as decimal degrees with 0 places. Take note of the Sample Output area, which provides examples of the current units.



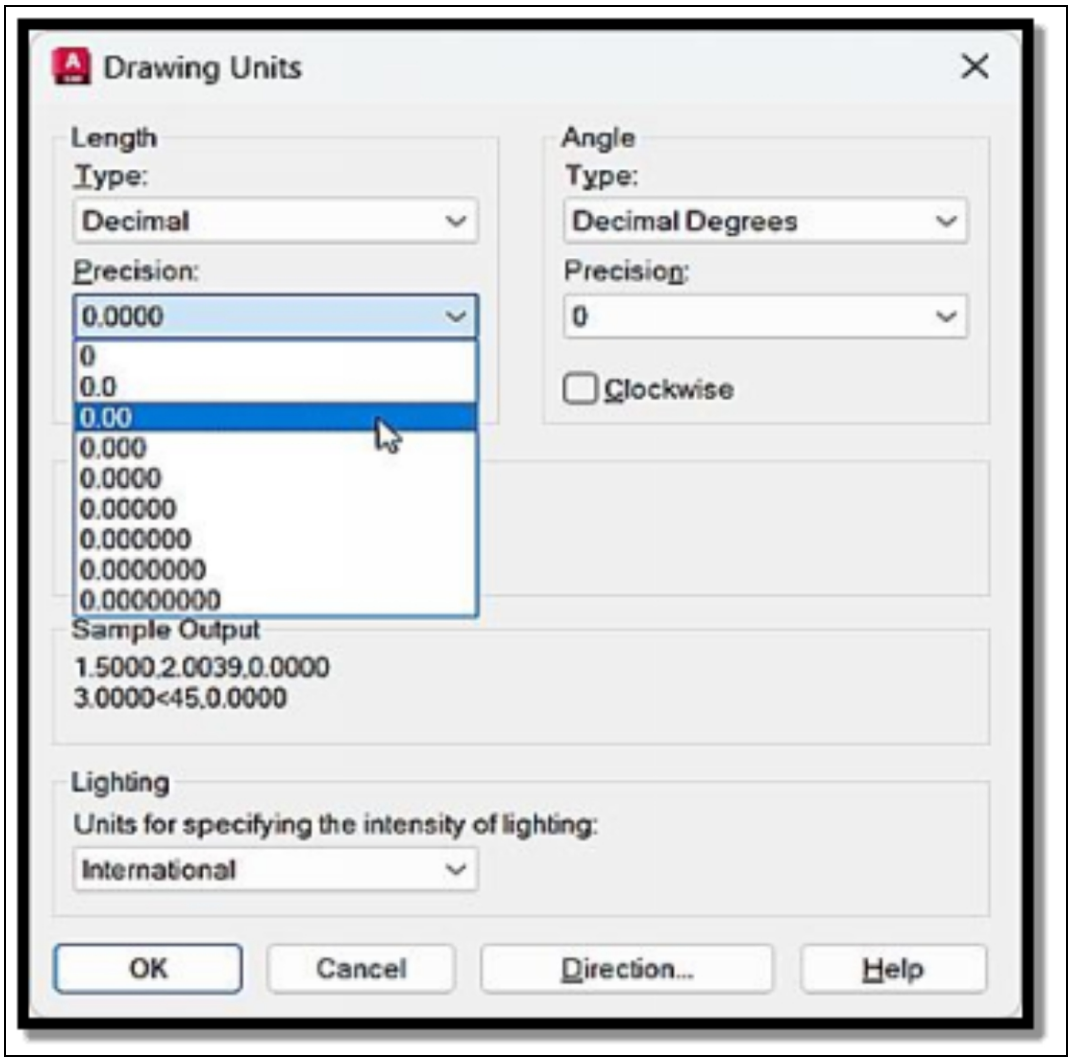
- In the Length panel of the dialog box, click the arrow located to the right.

**The drop-down list includes the following options:**

- Decimal
- Architectural
- Engineering
- Scientific
- Fractional

Architectural units in AutoCAD are used to display measurements in feet and fractional inches, such as 1'-3 1/2". Engineering units, on the other hand, display measurements in feet and decimal inches, like 1'-3.50". If you prefer a mixed-number format, you can use fractional units, which display measurements as 15 1/2. Lastly, if you're dealing with very large or very small numbers, AutoCAD's scientific units use exponential notation, such as 1.55E + 01. These formats are compatible with various units of measurement, except for engineering and architectural formats. Decimal mode is compatible with both metric and imperial units. When designing a house, architectural units are typically the preferred choice. If you are constructing a bridge, it would be beneficial to use engineering units. For mapping subatomic particles, it is advisable to utilize scientific units. Regardless of your specific application, modifying units in AutoCAD is a fundamental skill that can be utilized at any given moment. When starting a new drawing, it is crucial to select the correct units as part of your drawing practice.

- To change the setting, either select **Decimal** from the drop-down list or click outside the list. Next, adjust the precision setting to display decimals up to two places.
- To access the Precision drop-down list in the Length panel, simply click on the down arrow located next to the value 0.0000. The list displays a range of options from 0 to 0.00000000.




**Two-place decimals are often preferred due to their widespread usage.**

- Choose the option "**0.00**" from the drop-down menu.

The list is now closed, with **0.00** being the new precision for units of length, replacing **0.0000.** It is important to observe that the sample output has been modified to accurately represent the updated configuration. You can easily control the units for angular measurements and polar coordinates by adjusting the settings on the right side of the dialog box. When you access the Angle Type drop-down list, you will be presented with the following options: The default system in AutoCAD is set to standard decimal degrees, with zero decimal places. Measurements are taken counterclockwise, with zero degrees representing a straight line to the right (3 o'clock), 90 degrees representing a straight line upwards (noon), 180 degrees representing a line

to the left (9 o'clock), and 270 degrees representing a line downwards (6 o'clock). Do not modify these settings.

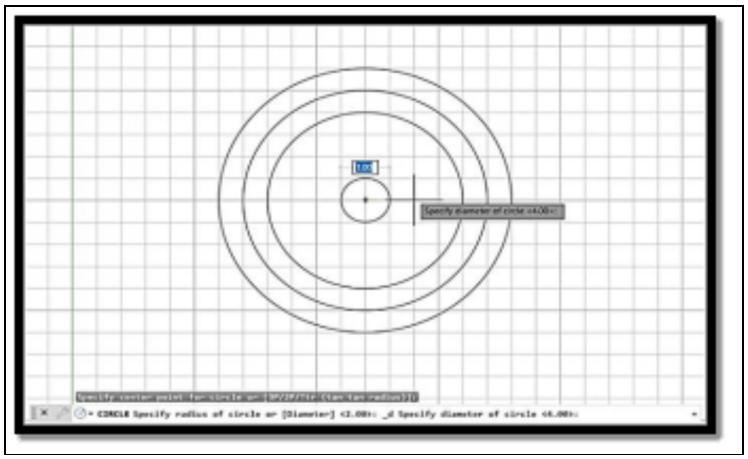
- Ensure that your length measurements are set to two decimal places and your angle measurements are set to zero decimal places.
- Click on the OK button to close the dialog box.

# How to Draw Circles by Specifying a Center Point and a Diameter

Afterward, you can easily create three additional circles by specifying their centers at the coordinates (6,5) and setting their diameters to 1, 1.5, and 2 respectively. This method for drawing circles is similar to the radius option, but there are some differences in how the rubber band and dynamic input display function.

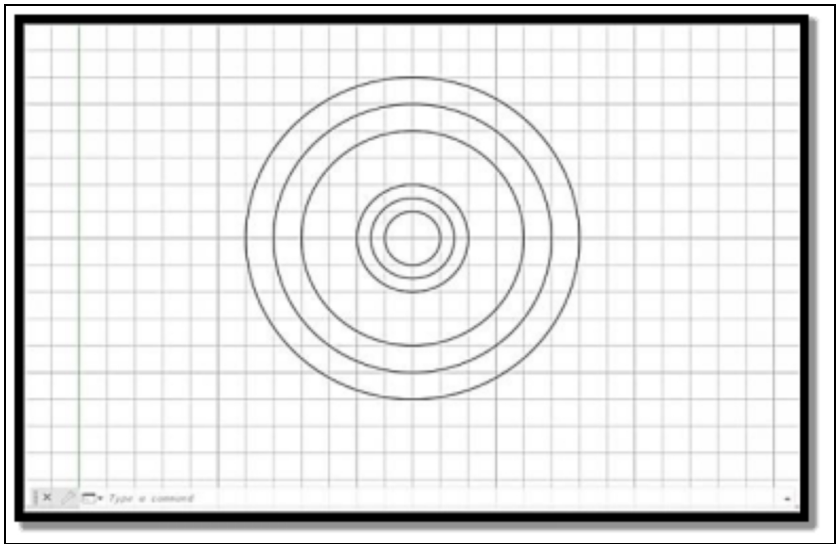
- Access the **Circle** drop-down menu located on the Draw panel of the Home tab. Then, select the **Center, Diameter option**.
- Indicate the center point located at (6,5).
- Adjust the position of the crosshairs away from the center point.

Observe that the crosshairs have been positioned outside the circle you are currently dragging within the drawing area. The reason for this is that AutoCAD is expecting a diameter, but the last point you provided was a center point. When measuring the diameter, it is important to start from the center point and extend outwards, as it is equal to twice the radius. Additionally, it is worth noting that the dynamic input display has promptly adjusted to the Diameter option and is currently displaying the circle's diameter. Manipulate the cursor in various directions relative to the center point to gain a better understanding of its functionality.



When you want to repeat a command in AutoCAD, keep in mind that simply pressing <Enter> or the spacebar won't repeat the option unless it happens to be the default option. By default, the command will use the Radius option instead of the Diameter option in this scenario.

- Set the diameter of the circle to 1.00 units. By now, you should have four circles in your design.
- Create two additional circles with diameters of 1.50 and 2.00.



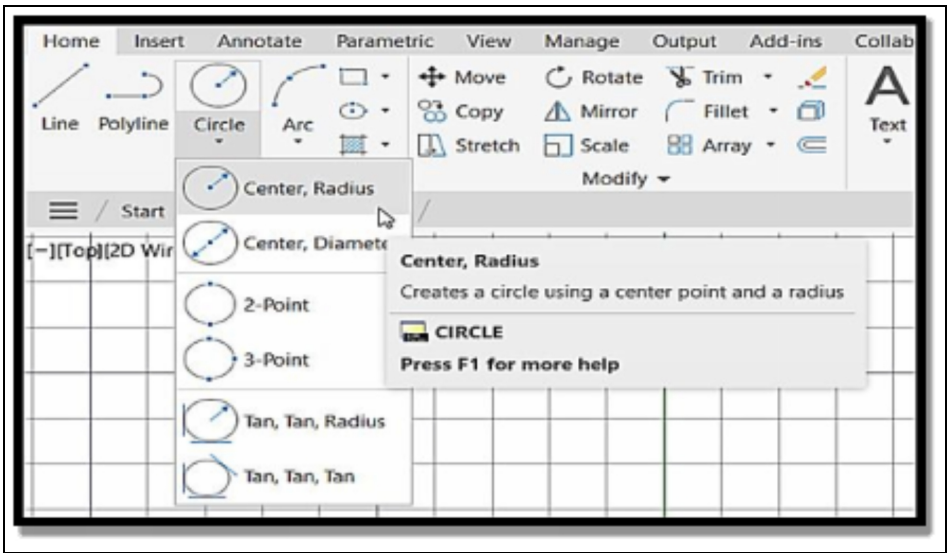
# How to Draw Circles by Specifying a Center Point and a Radius

**Here is a step-by-step guide on how to draw a circle:**

- To access the drop-down menu for the Circle tool, locate the arrow located below it on the Draw panel of the Home tab in the ribbon.
- Use the Center, Radius tool by clicking on it. Indicate the center point.
- Provide the desired radius value.
- There are various ways to draw circles in AutoCAD. You have the option to specify a center point and a radius, a center point and a diameter, two points that determine a diameter, three points on the circle's circumference, two tangent points on other objects and a radius, or three tangent points. These various options are conveniently located in a drop-down menu below the Circle tool on the ribbon. For this chapter, you will only be utilizing the initial two options.
- To start, create a circle with a radius of 3 and centered at the coordinates (6,5). Next, create two smaller circles that are centered at the same point. Afterward, you can remove them by using the ERASE command.

The tool displayed on the ribbon will vary based on the most recent selection made. When the last CIRCLE command option used was Center, Radius, the ribbon will display the Center, Radius tool. **For this scenario, executing Center and Radius commands can be done without the need to open the drop-down menu.**

- For optimal precision, it is recommended to set the grid to 0.50, the snap to 0.25, and the units to a two-place decimal.
- To access the drop-down menu for the Circle tool, locate the arrow below it on the Draw panel of the **Home tab** on the ribbon.

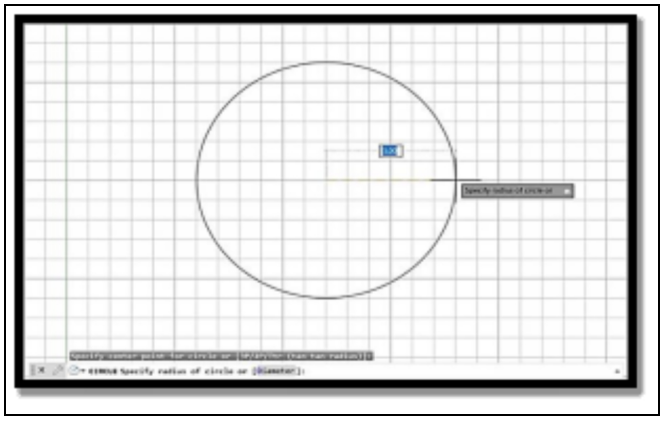



# Circle tool – drop-down menu

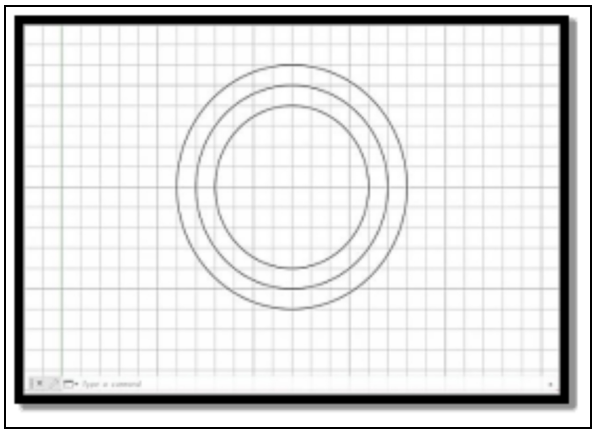
To access the drop-down menu, refer to the illustration provided. There are numerous drop-down menus available on the ribbon, providing convenient access to a wide range of **commands and options without cluttering up the ribbon interface. The center and radius options can be found at the top of the menu.**

- Select the **Center, Radius tool** from the drop-down menu. Provide the center point for the circle. You can choose between 3P, 2P, or Ttr (tan tan).
- Provide the center point coordinates (6,5) for the circle. When using AutoCAD, the software will ask you to input either a radius or diameter.
- Indicate the desired radius of the circle or, if you prefer, provide the diameter: When a coordinate value is entered or specified, AutoCAD automatically interprets it as a radius by default. The only remaining option is the diameter. Observe the rubber band and dragged circle as you move your cursor. To enable the dynamic input display, simply click the Dynamic Input button located on the status bar.
- In addition to the rubber band and circle, you may observe that the coordinate display now indicates polar coordinates. If the coordinate display is set to static mode, this information will not be visible.

- To ensure the Coordinates display updates in real-time as you move the cursor, simply click on it on the status bar. Keep in mind that when working with polar coordinates, you'll be dealing with both a length and an angle. For this scenario, the length refers to the radius indicated by the rubber band. Dynamic input is an excellent tool for creating circles in AutoCAD. You can easily determine the radius or diameter of the circle being drawn at the exact point you specify.
- Observe the dynamic input display or the coordinate display and indicate a point that is 3.00 units away from the center point.



**Here's a helpful suggestion:** To create multiple objects of the same type, you can use the **Add Selected option** found in the shortcut menu. To add a selected object, simply right-click on it to open the shortcut menu, and then choose the option "**Add Selected**". To add an object of the selected type, you can use the **ADDSELECTED command**. Create two additional circles by using the identical center point and radius option. The circles should be positioned at coordinates (6,5) and have radii of 2.50 and 2.00. To repeat the command, simply press the spacebar or click on the Center, Radius tool from the ribbon once more.



# How to Use Single-Point Object Snap

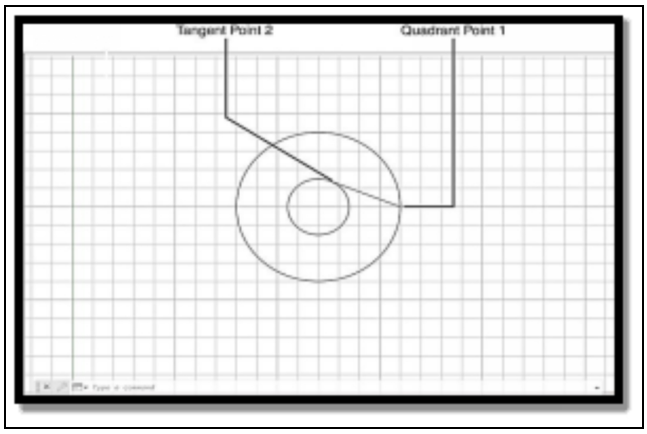
Now that you have a solid understanding of object selection methods, it's time to shift our attention back to drafting and drawing aids. In this section, you will find a brief overview of the highly efficient object snap feature and the convenient single-point object snaps. This feature allows you to snap to specific points on objects that have already been drawn, instead of relying solely on the coordinate system. With this feature, you can precisely define points that may be difficult to locate using the crosshairs or coordinate input. Object snap is a useful feature in AutoCAD that allows you to easily locate specific points on previously drawn objects. With object snap, you can quickly find geometrically definable points like midpoints or endpoints of lines.

**Here's a helpful tip for you.**

- Input a drawing command, such as LINE or CIRCLE.
- To perform the desired action, simply use the right-click function while simultaneously holding down either the **Shift or Ctrl key**.
- Choose an object snap mode from the shortcut menu.
- Select an existing object.
- When working with the circles you previously drew, you can easily connect them by drawing a line that is tangent to both circles. This can be done by utilizing the quadrant and tangent object snaps, which will ensure precise alignment.
- A quadrant refers to a specific point on a curved object, positioned at 0°, 90°, 180°, or 270° from a given point. It can also refer to the area enclosed between any two adjacent axes.

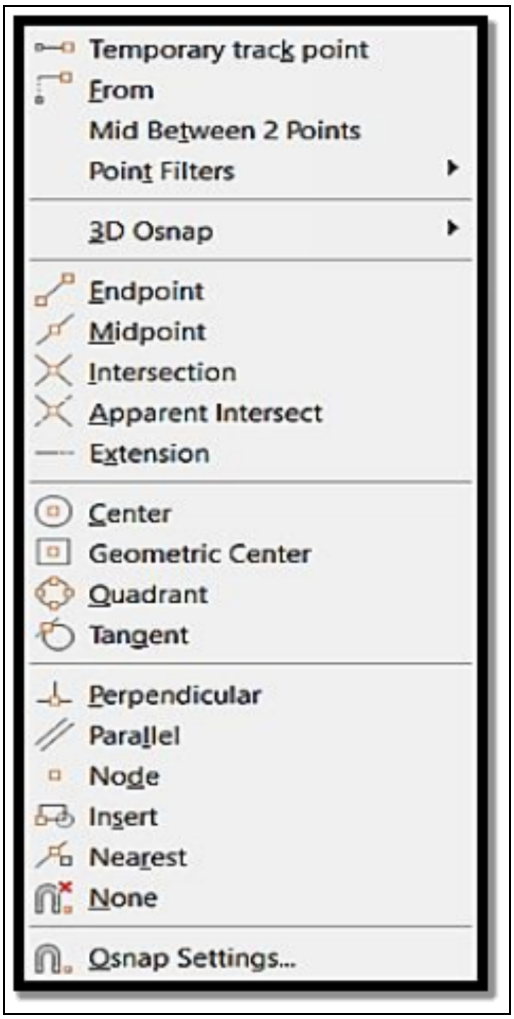
- A tangent is a line that intersects a circle at only one point, touching it at that point. It is perpendicular to the radius of the circle at that point.
- Make sure to disable the Object Snap and Dynamic Input buttons on the status bar for this exercise.
- At the command-line prompt, type in "**LINE**".

To create the desired geometric configuration, a line is drawn from the quadrant of one circle to the point of tangency with another circle. To achieve this, you will need to utilize the Tangent object snap feature. Locating the quadrant point is a breeze, as it falls on a grid snap point. However, pinpointing the tangent point proves to be a bit trickier, as specifying a point won't do the job. When using AutoCAD, you can easily select an object snap mode from the Object Snap shortcut menu when prompted for a point. This feature allows for precise and efficient drawing.

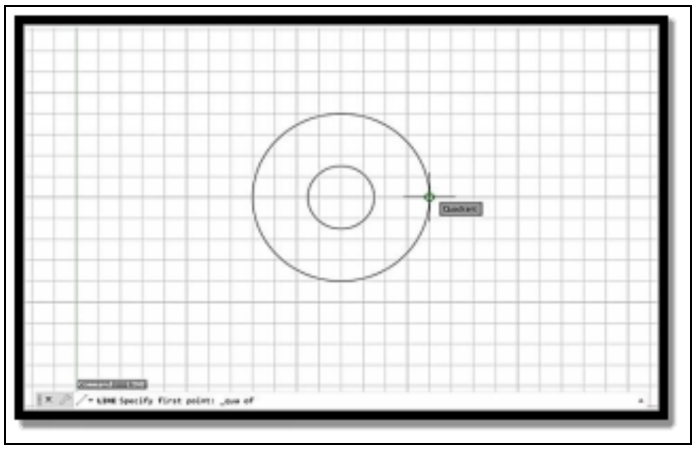



**To draw a line from a quadrant of a circle that is tangent to another, follow these steps:**

- When prompted to specify the first point, you can utilize a handy shortcut. Simply hold down either the <**Shift**> or <**Ctrl**> key and right-click.

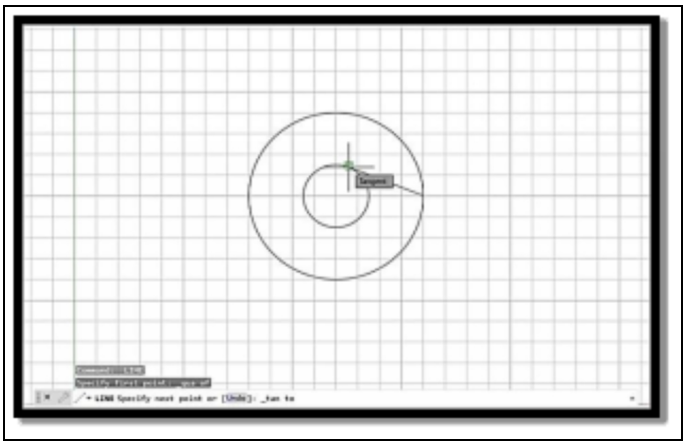



- Choose the **Quadrant option** from the shortcut menu. By using a Quadrant object snap, you can indicate to AutoCAD that you intend to select the starting point of the line. This method allows you to avoid directly specifying or entering a coordinate value.
- Position the crosshairs precisely over the two circles in your drawing. When navigating through the circles, AutoCAD conveniently highlights the eight quadrant points with a distinctive green diamond. The symbol used for object snap in AutoCAD is commonly referred to as a marker. Each type of object snap has markers of different shapes. When the cursor is placed near a marker for a brief period, a tooltip will appear, indicating the recognized object snap type.



# Use object snap to easily locate the quadrant point.

- Position the crosshairs close to the right quadrant of the larger circle, specifically at point 1.
- To activate the quadrant marker, simply click the left mouse button. When the green quadrant marker and snap-tip vanish, you'll notice a rubber band extending from the circle's center to the crosshairs' location. When you look at the command line, you will notice the prompt for specifying the next point. Use the Tangent object snap to precisely choose the second point.
- When you reach the Specify next point or [**Undo**] prompt, simply open the shortcut menu once more by using the combination of <**Shift**> + right-click. From there, you can easily select the Tangent option.
- Adjust the cursor to move upwards and towards the left, ensuring that the crosshairs are positioned close to the right side of the smaller circle.



After reaching the tangent area, the green tangent marker becomes visible. Once again, when the cursor is idle, a tooltip will appear displaying the object snap type.

# Locate the tangent point using object snap

- Activate the tangent marker and initiate the action by clicking the left mouse button. AutoCAD effortlessly identifies the tangent point and skillfully draws the line. Appreciate the immense capability of accurately determining the tangent point using this method.
- Press the **Enter key** to finalize the **LINE command**.

# How to Use ERASE Command

There are numerous editing methods available in AutoCAD. Essentially, most edit commands in AutoCAD can be used in two different sequences. These methods are commonly referred to as the verb/noun and noun/verb methods. The selection method where an edit command is initiated before selecting objects for editing. The selection method involves choosing an object before initiating an edit command. In this section, you will gain expertise in utilizing the verb/noun sequence followed by the noun/verb or "**pick first**" method, along with exploring the numerous options available for object selection.

**Here's a helpful suggestion: Here is a step-by-step guide on how to use the ERASE command with verb/noun selection:**

- Select the **Erase tool** located in the Modify panel on the Home tab of the ribbon.
- Choose the desired objects.
- Press the **Enter key** to execute the command.

# Verb/Noun Editing

- Before proceeding, ensure that you have the six circles in your drawing. To erase the two outer circles, you can use verb/noun editing. To delete two circles simultaneously, you can use this method.
- Select the **Erase tool** located in the Modify panel on the Home tab of the ribbon. When using the command line or dynamic input display, you will notice the prompt.
- Choose the desired objects: This prompt is commonly encountered in various edit commands and numerous other commands.
- Position your cursor, resembling a box referred to as the pickbox, above the outer circle. The circle is disabled and a red X is displayed.
- Click on the outermost circle to select it. The circle retains its gray color while the red x vanishes.
- Use the pickbox to precisely select the second circle as it moves closer to the center. The option should now be disabled.
- Use the right-click option or simply press the Enter key to execute the command. This follows the common verb/noun sequence found in many edit commands. After initiating a command and selecting the desired objects to form a selection set, it is necessary to press <**Enter**> or right-click to finalize the command. Now, it's time to remove the two outer circles.

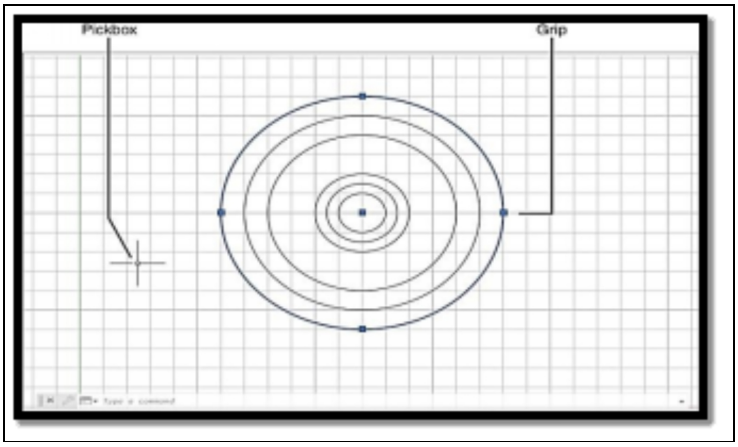
**Here's a helpful suggestion:** When using verb/noun edit commands, it is advisable to use the right button on your mouse instead of the spacebar or <**Enter**> key to complete the command. This is a helpful tip to keep in mind.

# Noun/Verb Editing

**Now, let's give the noun/verb method a try:**

- To undo the **ERASE command** and restore the circles, you can either click the Undo tool on the Quick Access toolbar or simply type 'u' followed by the Enter key. At the center of your crosshairs, you will see a square. This square is the pickbox, which you may remember from using the ERASE command with the verb/noun method.
- Hover your cursor over the outermost circle and click to make a selection. The circle has been highlighted in a vibrant shade of blue, perfectly complementing the overall aesthetics of your drawing area. The small blue boxes are commonly referred to as grips in AutoCAD. They can modify various objects in AutoCAD. Currently, you can disregard them.

Grips are strategically placed at geometric locations on selected objects in an AutoCAD drawing. You can achieve various forms of editing by simply clicking, dragging, and manipulating grips.




- Choose the second circle that is located on the outermost part using the same method. When the second circle is selected, it becomes highlighted and additional grips are displayed.
- To access the Erase tool, navigate to the Modify panel located on the Home tab of the ribbon. Simply click on it to activate the tool. In addition, you have the option to input "e" followed by pressing the **Enter key** or using the **Delete key**. The two outer

circles have been removed. The two outer circles have been removed once again. Upon observation, it is evident that there is minimal disparity between the two approaches. Several selection methods in the verb/noun method are not immediately noticeable but cannot be activated when objects are selected first. Take a moment to explore other object selection methods before giving the OOPS command a try.

**Here's a helpful suggestion:**

- Choose objects.
- To access the Erase tool, navigate to the Modify panel located on the Home tab of the ribbon. Simply click on it to begin using it.
- If you type "**oops**" and press Enter, you can observe the changes in the drawing area. If an error occurs during the erasure process, you have the option to retrieve your objects by using the "**oops**" command. Just like how "**redo**" is the opposite of "**undo**," "**oops**" is the counterpart of "**erase**." To undo the most recent ERASE command, regardless of any subsequent commands, you can use OOPS. This feature allows you to revert the action as long as you haven't executed another ERASE command in the meantime.

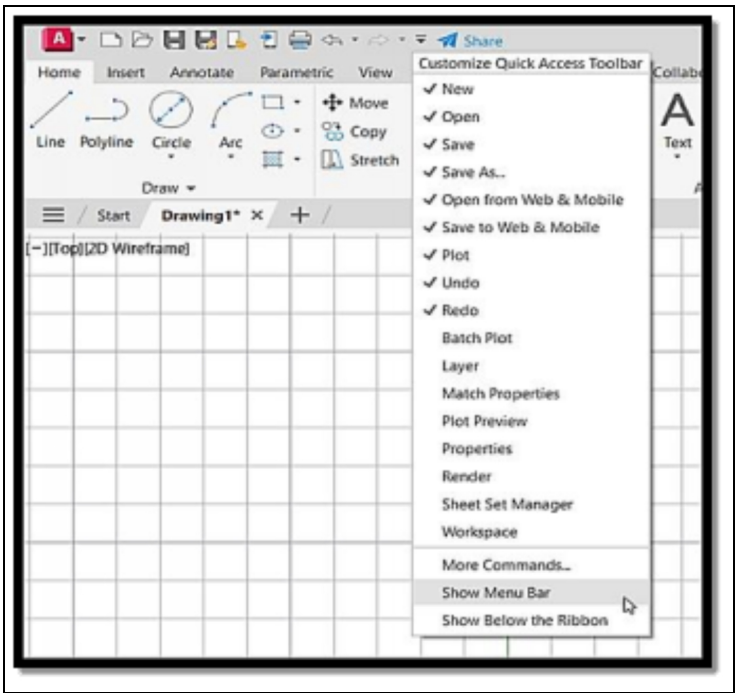
Another option is to use the U or Ctrl-Z commands to undo an ERASE action. It's important to note the distinction: U will undo the most recent command, regardless of what it was, while OOPS is specifically designed to work with ERASE and retrieve the previously erased objects. If you have already created other objects, you can use the OOPS command to retrieve previously deleted objects. On the other hand, if you were to use the U command, it would necessitate going back and reversing any recently created entities in the process.

# How to Customize Your Workspace

There are numerous methods available to customize your workspace according to your preferences or the requirements of a specific drawing. The Drafting & Annotation and 3D Modeling workspaces are typically adequate for most tasks, but making small adjustments can greatly enhance

efficiency. Here, you can implement two straightforward modifications that will enhance the efficiency of your workspace. Once the modifications are complete, simply save the updated workspace with a new name. From then on, you can effortlessly utilize it as a replacement for the Drafting & Annotation workspace whenever the need arises. In previous versions of AutoCAD, before the introduction of the ribbon and application menu, the menu headings were located at the top of the application window and would open pull-down menus, following the convention seen in other applications. The interface you're referring to is known as the menu bar. It's a convenient feature that can be easily incorporated into your workspace without taking up too much of your valuable drawing area. The **menu bar** is a collection of drop-down menus that are accessed through labels displayed horizontally on a bar. This bar is located below the application title bar and above the ribbon. The menu bar is not displayed by default. You can easily display it from the Quick Access toolbar.

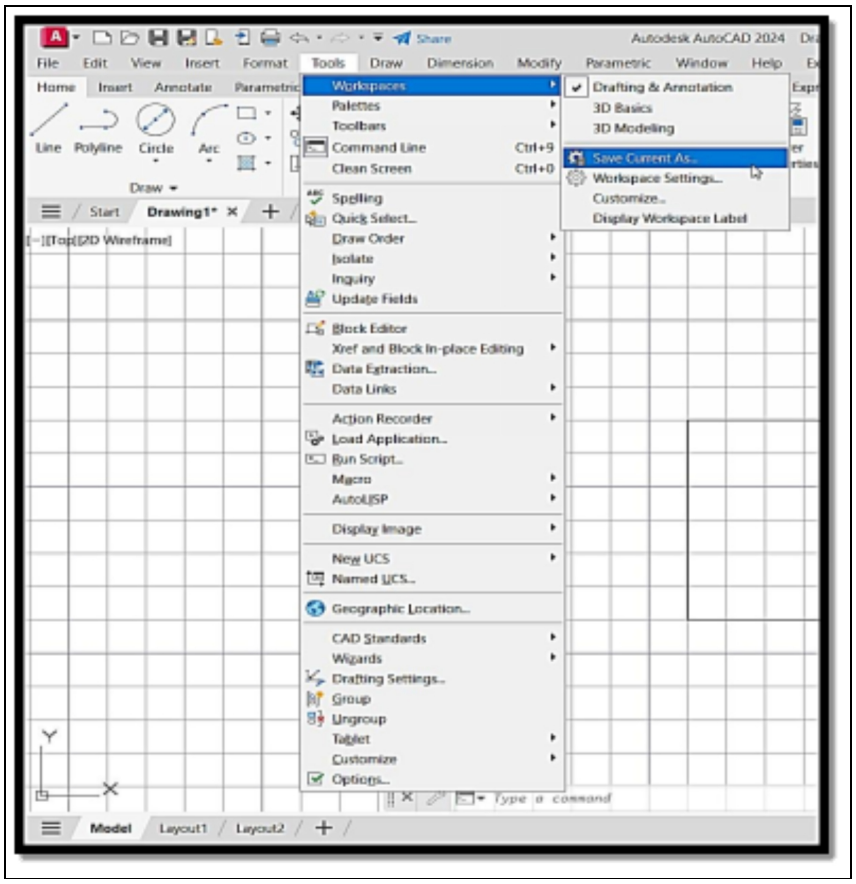
- To access additional options, simply click on the arrow located at the right end of the Quick Access toolbar.



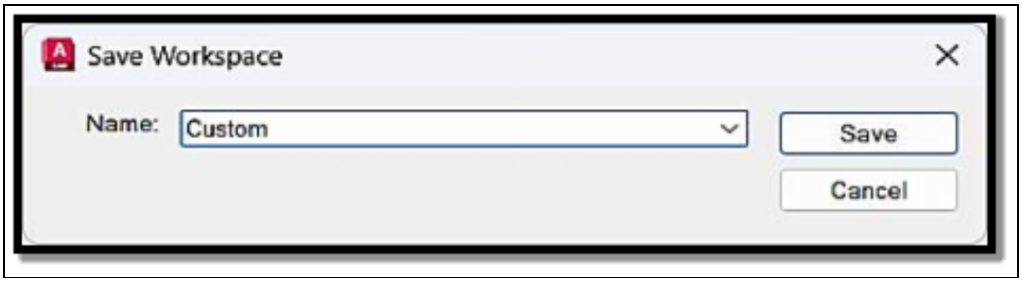

- To display the menu bar, simply select "**Show Menu Bar**" from the options. The menu bar is located at the top of the application window, positioned between the ribbon and the application title

bar. When you click on any of the headings such as File, Edit, View, Insert, and so on, a drop-down menu will appear. There are a variety of commands and procedures that can be accessed from these menus. To disable the menu bar, you can follow the steps in reverse.

- Once more, simply click on the arrow located at the right end of the Quick Access toolbar.
- To access the Layer menu, simply select it from the menu bar. A new feature has been introduced to the Quick Access toolbar - the Layer drop-down list. The Layer drop-down list on the Layers panel of the Home tab remains consistent across all ribbon tabs, providing easy access regardless of the current tab. Once you have the menu bar and Layer drop-down list visible, you can easily save these modifications to the current workspace by giving it a new name. With the menu bar open, you can easily save your workspace.
- To save the current workspace, go to the menu bar and choose **Tools**, then select **Workspaces**, and finally click on **Save Current As**.




# Saving the existing workspace




- Press the "**Enter**" key after typing "**Custom**".

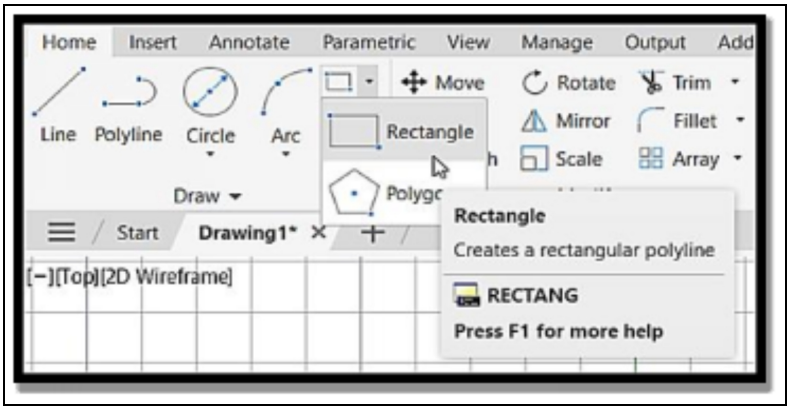
To confirm that your Custom workspace has been successfully saved, go to the menu bar and select **Tools** > **Workspaces** once more. A new workspace called "**Custom**" has been added to the list and is visually emphasized to indicate that you are currently working in this specific workspace. You can easily switch back to the Drafting & Annotation workspace by selecting it from the Tools menu on the menu bar or the Workspace Switching button

on the status bar. Once you perform this action, the menu bar and the Layer drop-down list on the Quick Access toolbar will no longer be visible in the application window.

# How to Use RECTANG Command

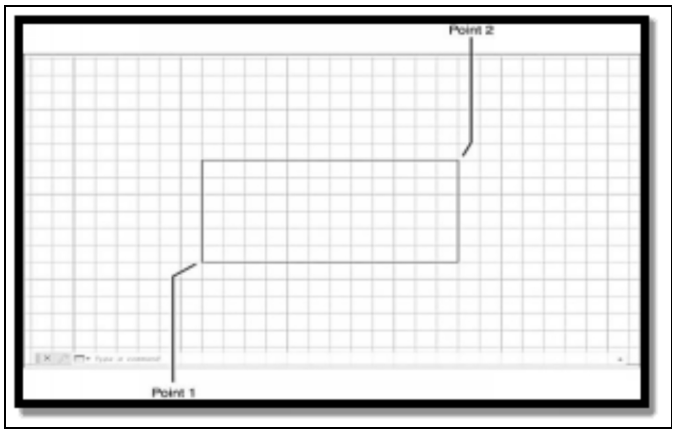
Once you have mastered the art of defining object selection windows, the RECTANG command will become second nature to you. **Creating a rectangle using the RECTANG command is similar to drawing a selection window for objects.**

- Select the **Rectangle tool** from the drop-down menu located on the Draw panel of the Home tab.
- Provide the coordinates for the first corner point.
- Indicate the point at the diagonal corner.
- Before commencing this exercise, it is advised to remove all objects from your drawing.
- Enable **Snap Mode** and choose **Dynamic Input** from the status bar.
- Select the Rectangle tool from the drop-down menu on the Draw panel of the Home tab.




**When using AutoCAD, the software will prompt you to specify a corner point.**

- Feel free to disregard the options and continue with the default settings.
- For the first corner point, input the coordinates (3.00,3.00)




- A rectangle with dimensions of 6 × 3.

AutoCAD is requesting input for an additional point.

- Specify the coordinates of the other corner point or select an option from Area, Dimensions, or Rotation:

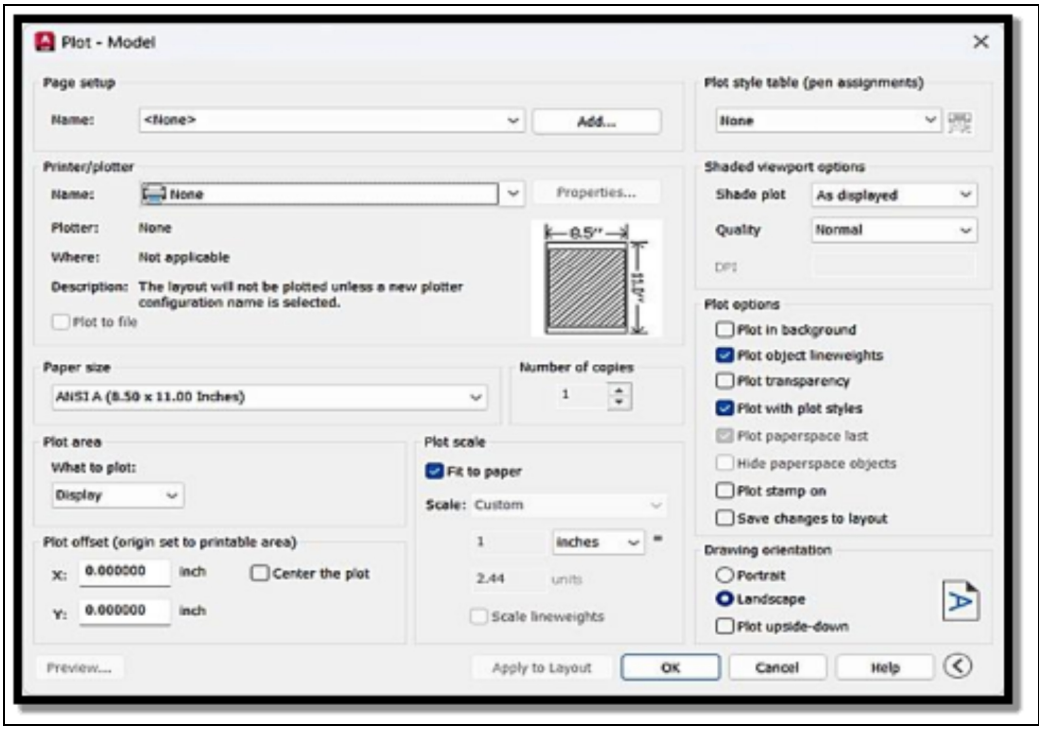
**Here's a helpful suggestion:** Observe the distinct behavior of the coordinate display and dynamic input tooltip display once the RECTANG command is executed. The coordinate display in AutoCAD remains consistent by showing absolute coordinates that are relative to the display grid. Dynamic input displays the absolute coordinates to the initial corner point of the rectangle.

- To create a 6 × 3 rectangle, simply specify a second point at (9.00,6.00).

# How to Plot or Print a Drawing

The printing and plotting capabilities of AutoCAD are highly advanced and intricate. In this section, you will learn how to easily create a plot by transferring your model space objects to a drawing paper with minimal changes to the plot settings. Various types of plotters and printers have distinct operational methods, but the following exercise will help you achieve consistent outcomes. Assuming there is no need to alter devices or make any significant changes to the configuration details. To define a plot area and scale it to fit the paper size in your plotter or printer, you can use a window selection.

- Access the Plot tool by clicking on it in the Quick Access toolbar.




# Setting up a Printer

First, it is essential to select a plotter or printer for your project. The panel located just below the top panel is dedicated to Printer/plotter settings. **Check if the name of a plotting device is visible in the Name List box.**

- If the list box does not display any options, simply click the arrow on the right and choose a plotter or printer from the available selections. If you're uncertain about which plotter to use, you can opt for the AutoCAD PDF (General Documentation).pc3 device, which should already be available.
- Take a look at the Plot scale panel located either in the lower right of the reduced dialog box or in the lower middle of the expanded dialog box. Find the Fit to paper check box. When your plot scale is set to fit your paper, AutoCAD will automatically plot your drawing at its maximum size, taking into account the paper size and window you specify.

- To ensure that Fit to Paper is enabled, simply click on the box if it is not already checked. Below the Plot scale, you will find edit boxes displaying numbers such as 1 inch = 2.45. Currently, the plot area is determined by the dimensions of your display area, making it difficult to access specific measurements. When creating a plot area that is slightly smaller using a window, the scale numbers will automatically adjust. When utilizing the Scale option instead of Fit to Paper, you will have easy access to these numbers. You can either manually input them or choose from a provided list.

# Plot Area

Take a moment to direct your attention to the panel situated at the lower left, specifically labeled as the Plot area. The "What to plot" drop-down list provides options for selecting the desired area of the drawing to be plotted. You can choose to plot what is currently displayed, the entire extent or limits of the drawing, or a specific window that you define. The Window feature in AutoCAD enables you to plot specific sections of a drawing by defining the desired area in the usual manner. The size and placement of the plot in AutoCAD are determined by the window that you define.

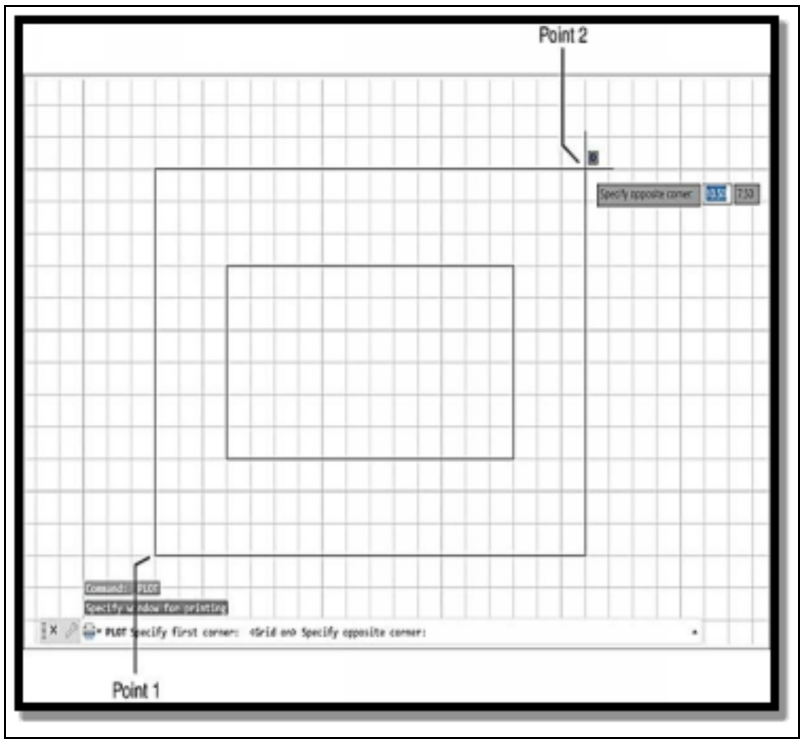
- To select the "**Window**" option, simply click the arrow located on the right side of the "**What to plot**" drop-down list. Temporarily, the Plot dialog box vanishes, granting you access to the drawing. When using AutoCAD, the software will prompt you to select a point:

**Indicate the desired window for printing. Provide the coordinates for the first corner:**

- Indicate the coordinates (1.50,1.50) for the point.

**AutoCAD provides prompts: Provide the coordinates for the diagonal corner:**

- Indicate the coordinates (10.50,7.50) for the point.



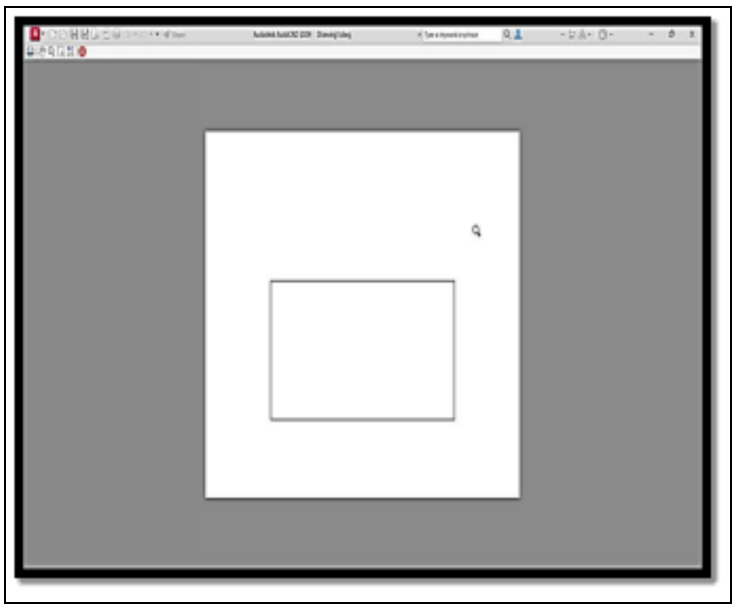

Once the second point is specified, the Plot window will reappear in AutoCAD.

# Plot Preview

Understanding the importance of plot preview is crucial for achieving optimal results in plotting and printing. Understanding plot settings is crucial to avoid wasting time and paper when printing drawings. It is highly recommended to preview your drawings before printing to ensure optimal results. AutoCAD offers two types of previews that provide a clear representation of how your drawing will appear on paper. Even before accessing the full preview, a partial preview is conveniently available on the right side of the Printer/plotter panel.

It provides a visual representation of the plotting area to the paper size but does not display the actual plotted drawing. The preview image will update dynamically as you make adjustments to various plot settings, including plot area and paper size. When you click on the Preview button, you can access the full preview which provides a comprehensive view of the drawing on a sheet of drawing paper. However, it does not provide specific details about the effective drawing area available in the partial preview.

- To access the preview feature, locate the button positioned at the bottom left corner of the dialog box and click on it.



There are additional features available in the Plot dialog box, allowing you to make extensive use of full and partial plot previews while adjusting plot settings. It is recommended to develop the practice of using plot preview at this stage. If you're experiencing any issues with the outcome not meeting your expectations, rest assured that you can rectify them in no time.

- Use the <**Esc**> key to navigate back to the Plot dialog box.
- To save your settings, including your plotter selection, click on the **Apply to Layout** button. You are now prepared to initiate the plotting process.
- To plot or print the rectangle, ensure that your plotter is ready and then click OK. Alternatively, select the **Cancel option**.

# Important

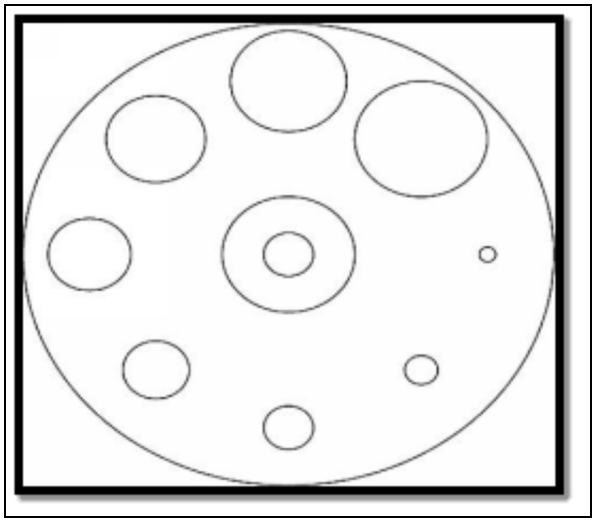
- This document does not cover paper sizes. Ensure that your plot settings are accurately aligned with the paper in your printer or plotter.
- When you click OK, the drawing information will be sent for printing. Observe the plotter in action with ease. If you

encounter any circumstances that require cancellation, simply click on the Cancel option.

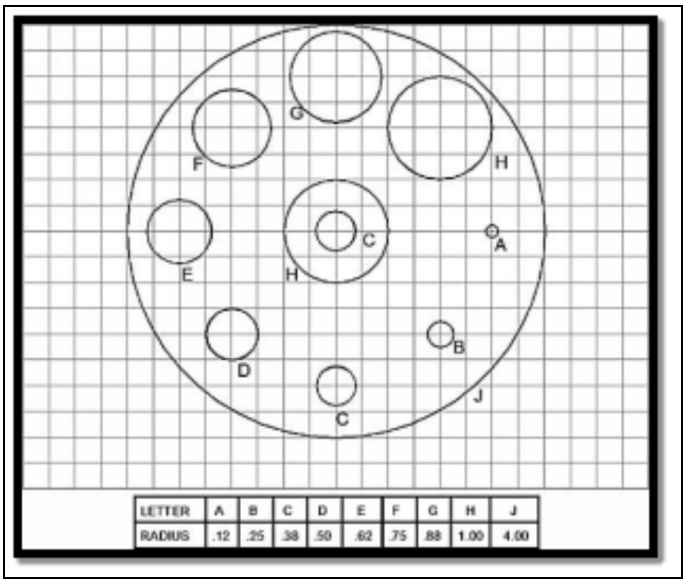
# Chapter Drawing Projects

- **Drawing 2-1: *Aperture Wheel* [BASIC]**

This drawing provides an opportunity to enhance your skills in creating circles using the center point and radius method. When the snap is set to 0.25, you can easily draw certain circles by simply pointing and dragging. Some circles may have radii that are not aligned with a snap point. To draw these circles, simply input the desired radius.



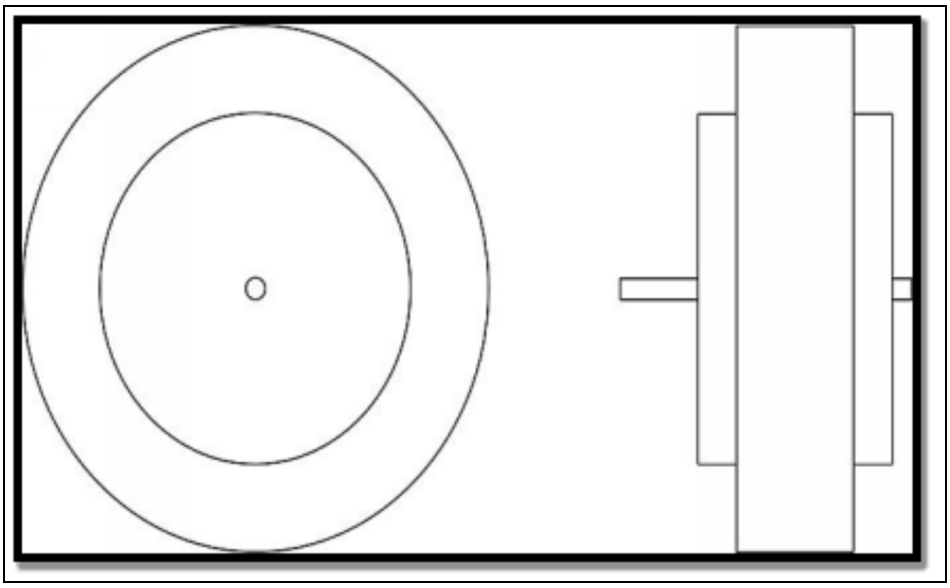
**Here are some suggestions for your drawings:** Set the grid spacing to 0.50 units and the snap spacing to 0.25 units. To create this drawing, it is recommended to start by drawing the outer circle. After that, proceed to draw the two inner circles (H and C). All of these are centered at the coordinates (6.00, 4.50). Start at circle A and proceed in a clockwise direction, ensuring that each circle is properly centered. It is important to observe that there are two circles labeled C and two circles labeled H. Both circles labeled with the same letter have identical dimensions. Keep in mind that AutoCAD allows you to input any value of your choice, resulting in a highly accurate graphical representation. However, it is not always possible to accurately indicate the specific point you desire through physical gestures. Typically, it is more time-saving to input specific values rather than disabling snap or modifying its configuration for a limited number of objects.



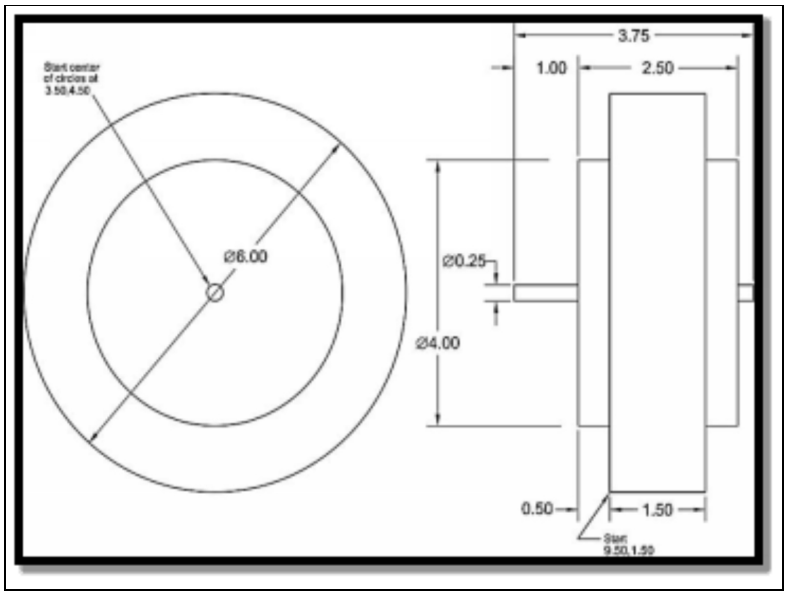

- **Drawing 2-2: Center Wheel [BASIC]**

This drawing provides an opportunity to seamlessly integrate lines and circles while utilizing the center point and diameter method. Additionally, this provides you with valuable hands-on practice working with smaller objects, a more intricate grid, and a more precise snap spacing.

**Here's a helpful suggestion:** It is crucial to ensure that the snap is set using three decimal places (0.125) to align the grid with a multiple of the snap (0.25 = 2 × 0.125), even if the units are set to display only two decimal places. AutoCAD displays precise rounded coordinate values, such as 0.13, while ensuring accurate alignment of the graphics. Experiment with adjusting the snap settings to values like 0.13 or 0.12, rather than 0.125, to observe the issue firsthand.

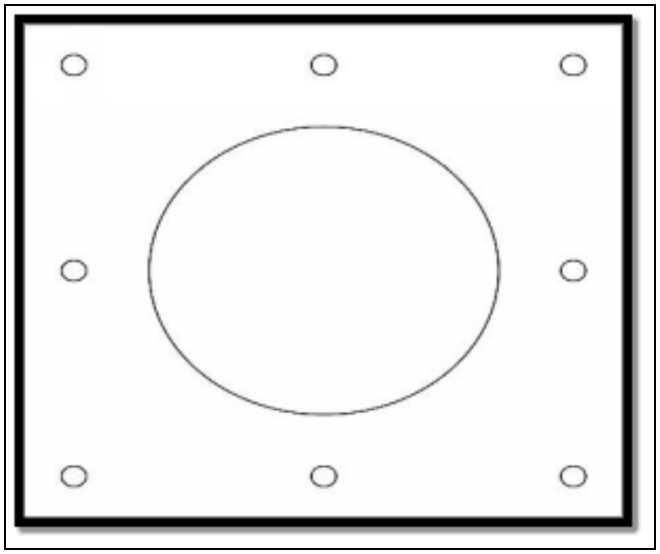


**Here are some suggestions for your drawings:** Set the grid spacing to 0.25 units and the snap spacing to 0.125 units. Aligning the two views of the center wheel is crucial, as it ensures the snap setting is properly utilized. Observe the coordinate display while you work and familiarize yourself with the reduced range of motion. Optimizing the sequence of actions greatly simplifies the completion of this drawing. To ensure proper alignment, it is recommended to begin by drawing the front view. Use circles with diameters of 0.25, 4.00, and 6.00. Next, utilize these circles to accurately position the lines in the right-side view. Ensure that the circles in the front view are precisely positioned at coordinates (3.50,4.50) for optimal alignment. The lower left-hand corner of the 1 × 1 square is positioned approximately at (4.50,1.50). Take note of the (Ø) symbol, which signifies that circles are specified by their diameter dimensions.



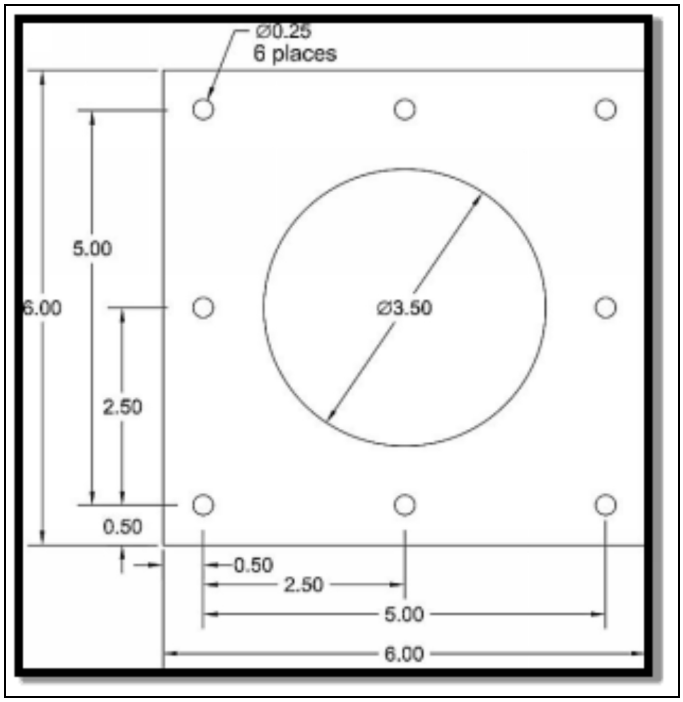

- **Drawing 2-3: Fan Bezel [INTERMEDIATE]**

You should find this drawing quite straightforward by now. Adjust the grid to a value of 0.50 and enable snapping at 0.125, as recommended. This will ensure that everything aligns perfectly.



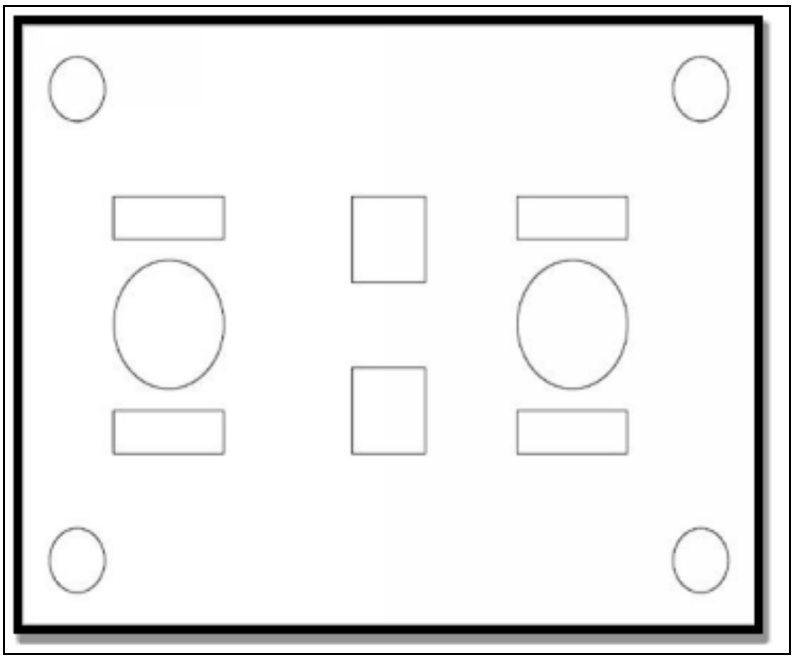
**Here are some suggestions for your drawings:** Set the grid spacing to 0.50 units and the snap spacing to 0.125 units. Take note of the diameters provided for the circles, which are indicated by the diameter symbol (Ø). For optimal centering of the drawing within the drawing area, it is recommended to begin at the lower-left corner of the square, preferably

around the point (3.00,2.00). Ensure precise alignment of the large inner circle within the square.

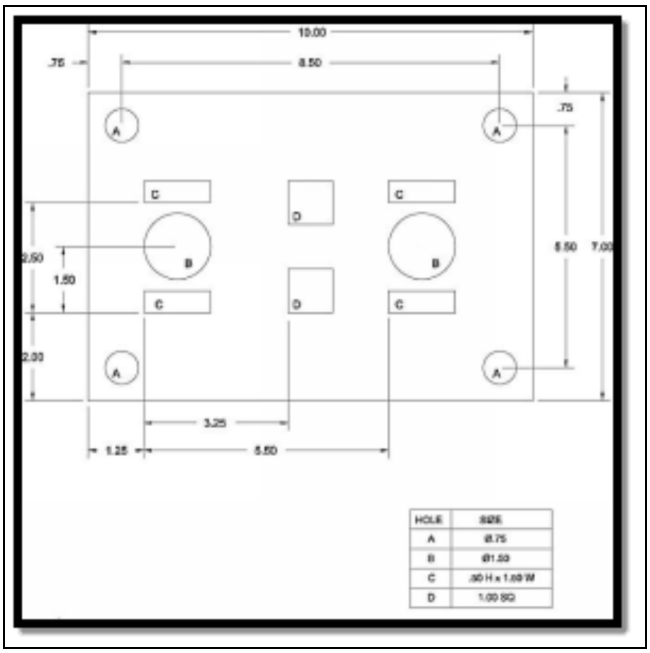



- **Drawing 2-4: *Switch Plate* [INTERMEDIATE]**

This drawing bears resemblance to the previous one, however, the dimensions pose a greater challenge, and several crucial points do not align with the grid. Mastering the use of grid and snap points, as well as the coordinate display, is essential for honing your skills in AutoCAD. The dimensions on the drawing itself indicate the placement of these smaller figures.

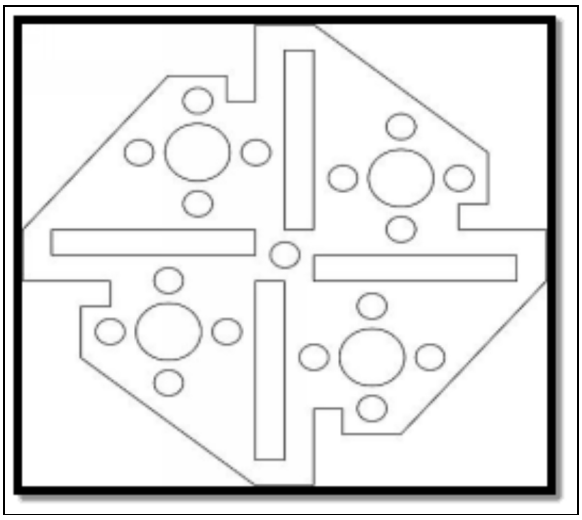


**Here are some suggestions for your drawings:** Set the grid spacing to 0.50 units and the snap spacing to 0.25 units. To achieve this drawing, enable the **Ortho** or **Polar** snap feature. Positioning yourself near the coordinates (1,1) is a good strategy to begin working in the drawing area.

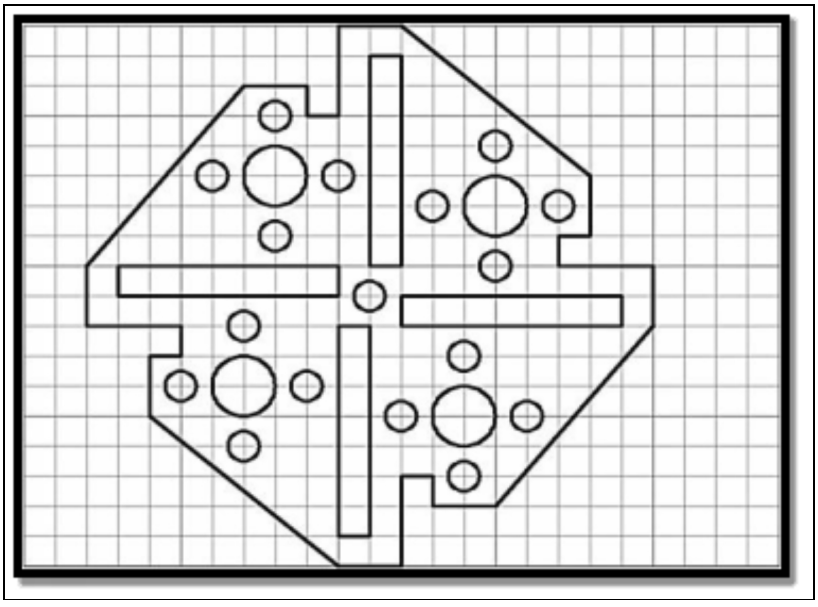



- **Drawing 2-5: Gasket [ADVANCED]**

Practice creating simple lines and circles by using the grid and snap features. The circles in this drawing have a diameter of 0.50 and 1.00. When the snap is set to 0.25, the radii will align perfectly with the snap points. Drawing circles in AutoCAD is a breeze. Simply drag the circle to display the radius or enter the diameter manually.



**Here are some suggestions for your drawings:** Set the grid spacing to 0.50 units and the snap spacing to 0.25 units. An optimal approach for completing this drawing would involve initially sketching the outer lines, followed by the inner lines, and finally the circles. It is important to observe that all endpoints of the lines align perfectly with the grid points, indicating that they are precisely on a snap point.



OceanofPDF.com

# CHAPTER SIX
# COPIES, TEMPLATES, AND ARRAYS

Here, you will discover some valuable techniques that can significantly enhance your productivity. If you're tired of repeatedly setting up layers, units, grids, snaps, and scales for every new drawing, then keep reading. Get ready to master the art of creating your very own drawing templates. Templates in AutoCAD allow you to easily start new drawings using setups that you have saved from previous sessions or choose from a range of predefined setups provided by the software. Furthermore, you will gain the expertise to modify the drawing grid by using the LIMITS command. Additionally, you will acquire the skills to efficiently duplicate, relocate, and arrange objects, eliminating the need for redundant drawings. Start by using the LIMITS command to modify the limits as you create your initial template.

## How to Set Drawing Limits

**Setting drawing limits is an essential skill in AutoCAD. Here's how you can do it:**

- Enter "**Limits**" at the command line. Provide the coordinates for the lower-left corner. Provide the coordinates for the upper-right corner.
- To execute the command, press the letter z followed by the **Enter key**. Then, press the letter a and hit Enter, or choose the "**All**" option from the command line.
- You have adjusted the density of the drawing grid multiple times, consistently within the confines of a 12 × 9 space. Now, you will learn the process of setting new limits to accurately represent different sheet sizes or any other desired space. However, before we proceed, let's briefly discuss the concept of model space and paper space.

# Model Space and Paper Space

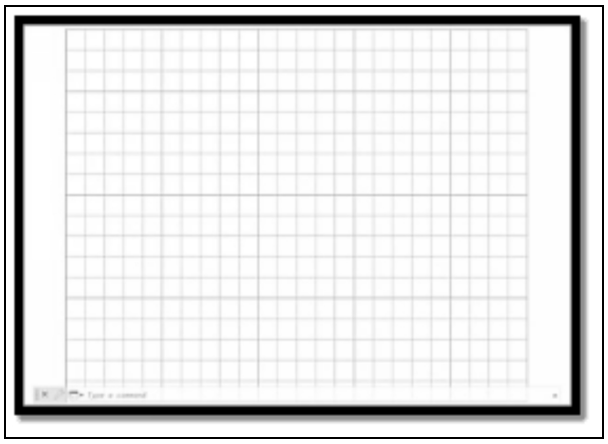
Model space is a fundamental concept in AutoCAD that encompasses the virtual environment where users can effortlessly create and modify objects. Objects in model space are always drawn at a 1:1 scale, meaning that 1 drawing unit represents 1 unit of length in the real world. In paper space, drawing units are used to represent the length units on a physical piece of paper, serving as an alternative to model space. When you start using AutoCAD's layout feature, you will come across paper space. A layout functions as a superimposed layer on your drawing, allowing you to define the sheet size, scale, and other paper-related settings. Layouts enable the creation of multiple views of the objects in the model space. When working in AutoCAD, it's important to understand the concept of limits. In this software, limits refer to the two points that establish the outer boundary of the drawing area within a specific drawing. The points are determined by ordered pairs in a Cartesian coordinate system, with the initial point representing the lower-left corner and the second point representing the upper-right corner of a rectangular space. A layout is a 2D representation that shows how a drawing will appear when it is plotted. Layouts are typically created in paper space, where viewports are used to position and scale objects drawn in model space for plotting purposes. In this exercise, you will learn how to modify the model space grid to match various drawing sheet sizes. This step is not required in subsequent exercises. When using AutoCAD, it is possible to easily adjust the scale of your drawing to perfectly fit any desired drawing sheet size during the plotting process. When considering model space limits, it is important to base them on the size and shape of objects in your drawing, rather than the paper used for plotting. **To start, create a new drawing and adjust the limits from an Architectural A-size sheet (12 × 9) to an Architectural B-size sheet (18 × 12).**

- To open the template list, you can either click the **New tool** from the Quick Access toolbar or click the down arrow next to **New** on the Start tab. You will be directed to either the Select Template dialog box or the Template drop-down list. At this stage, it is advisable to continue using the acad.dwt template. After creating

your template, it will be displayed in the dialog box or list alongside the existing ones.

- Choose the acad.dwt drawing template in the Select Template dialog box and click **Open**, or select acad.dwt from the drop-down list.
- Press the "z" key on your keyboard and then hit the **Enter key**.
- Press the **Enter key**. Up until now, you've been using the default settings that show the grid beyond the limits. To enhance your understanding of the limits, adjust the grid display to only cover the necessary areas. Afterward, you revert to the default configuration.
- To access the Snap Settings, simply right-click on the **Snap Mode button** located on the status bar. The Drafting Settings dialog box is displayed, with the Snap and Grid tab visible. In the lower right corner of this tab, you will find a panel called Grid Behavior.
- To disable the grid from extending beyond the limits, go to the Grid Behavior panel and deselect the option.
- Click the OK button.

**The grid is currently being displayed within the limits of (0,0) and (12,9).**



# How to Create and Save a Drawing Template

To create a custom template in AutoCAD, simply design a drawing with your desired settings and save it as a template. This will allow you to start new drawings with your preferred configurations effortlessly. **Here is a step-by-step guide on how to create a template:**

Set up layers and customize settings such as Grid, Snap, Units, Limits, Itscale, and more according to your preferences. Save the drawing as a template file in AutoCAD.

- **Modify the existing drawing according to the following instructions:**

Grid spacing is set to 0.50 units and the coordinate display is turned on. Snap is set to 0.25 and LTSCALE is set to 0.5. Set the units to a two-place decimal and adjust the limits to (0,0) and (18,12). Make sure to display objects beyond the limits.

# NOTE

Ensure that no elements are left in your drawing, as they will appear as part of the template whenever you create a new drawing. For certain applications, this feature can be highly advantageous.

- Set up layers with corresponding colors, line types, and configurations.

Keep in mind that you have the flexibility to modify your template whenever you need to. Creating the layers Dim, Hatch, and Text, even if they are not currently used, can be a time-saving and beneficial practice for your template in the future. Now, your drawing is prepared for saving as a template, which will be the main focus of the upcoming exercise.

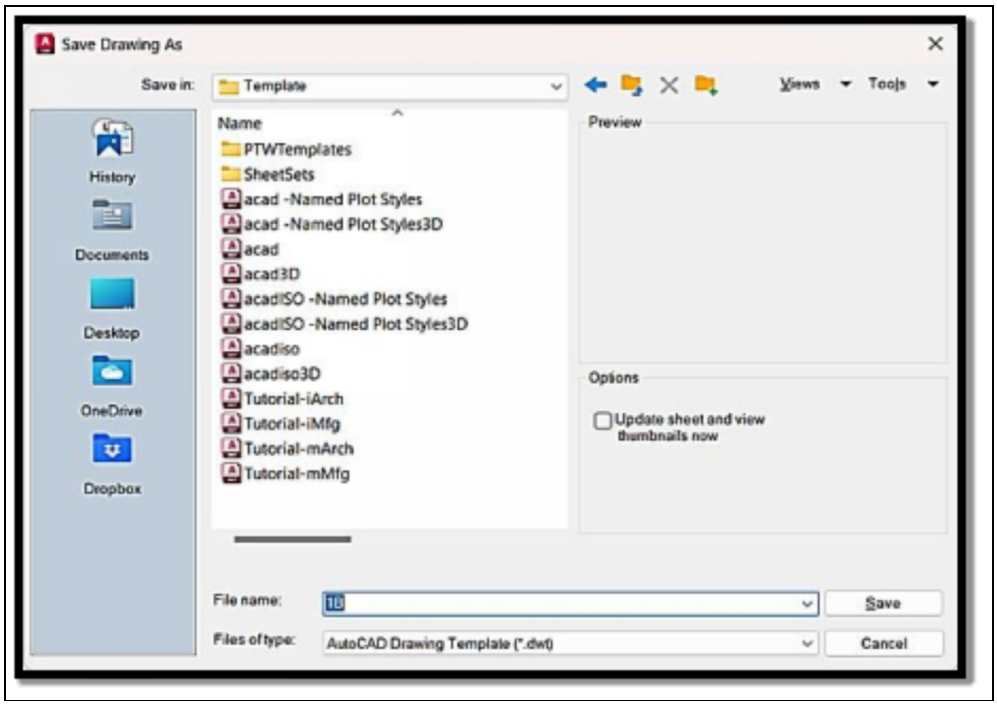
# How to Save a Drawing Template

When you save a drawing, it can be transformed into a template. Template files are typically assigned a .dwt extension and are stored in the designated Template file folder. When multiple users share a computer at different times, altering AutoCAD settings can lead to confusion. If your instructor or CAD manager has specific preferences, they may not want you to save a template file or may want it stored in a different location or under a different name.

- Make sure you're in the drawing from the previous exercise. Ensure that all modifications to the drawing are executed following the previously specified instructions.
- To save your work, simply click on the **Save As tool** located in the Quick Access toolbar.

# Note

The AutoCAD program comes with a range of templates that feature standard sheet sizes, complete with title blocks, borders, and predefined plot styles. These are incredibly user-friendly and efficient. Currently, they can lead to confusion as they are designed with paper space in mind and automatically place you in a paper space layout.

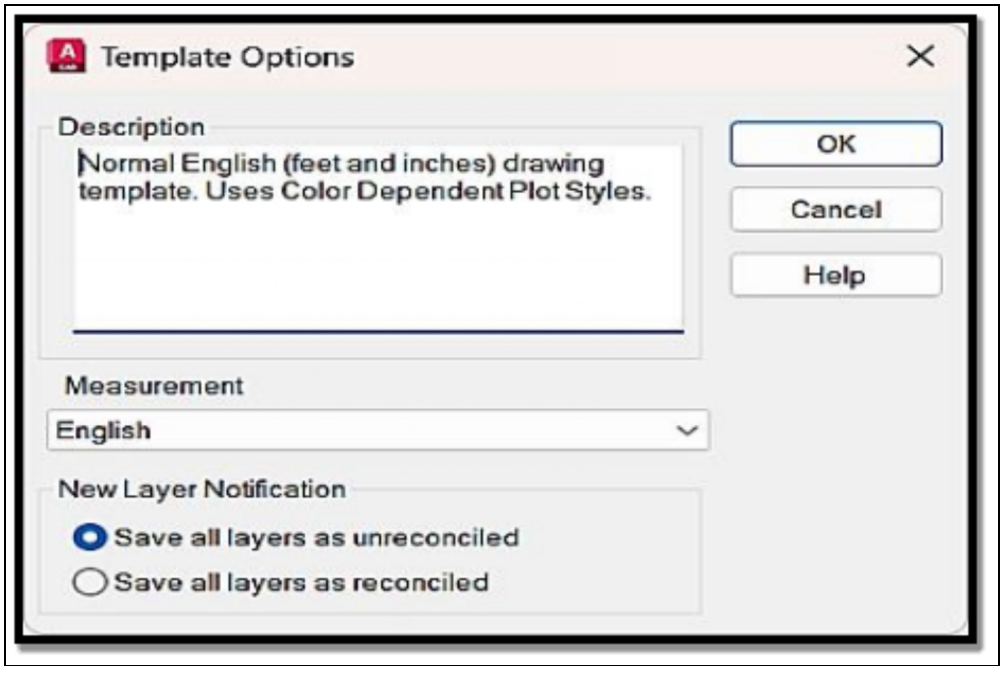



The File name edit box displays the name of the current drawing. When a drawing is not named, it is automatically assigned a default name such as Drawing1, Drawing2, and so forth. Located beneath the File name edit box, you will find the Files of type list. **This list provides various options for saving your drawing.**

- To access the Files of type list, simply click on the arrow located to the right of the list box. By selecting this option, you will be able to access a comprehensive list of file types.
- Choose the **AutoCAD Drawing Template** (*.dwt) file from the provided list. This will also automatically create the Template folder. Here is a list of commonly used templates that are often encountered when starting a new drawing. AutoCAD provides a wide range of templates that can be highly beneficial for your future needs. Now, it is crucial to master the art of creating your designs.
- If needed, simply double-click on the edit box for the file name. Enter "**1B**" as the new name. If multiple users share your computer, it might be helpful to include your initials to distinguish this template from your own.

**Here's a helpful suggestion:** To ensure that your template file appears at the top of the Files list in AutoCAD, it is helpful to begin the file name with a number. This way, it will be listed before the acad and ANSI standard templates that are included with the program. In the alphanumeric sequence, numbers come before letters. This means that your numbered template file will be positioned at the top of the list, saving you the hassle of scrolling down to locate it. ANSI standard refers to a variety of guidelines and standards that have been developed and disseminated by the American National Standards Institute. After entering a name for your drawing template in the File name edit box and ensuring that AutoCAD Drawing Template (.dwt) is selected in the Files of type list, you can proceed to save your work.

- Save the file by clicking on the Save button.
- Proceed by selecting the OK button within the Template Options dialog box. The drawing template has been successfully created. Now, it's time to create a new drawing using the template and observe its functionality.




# Take note

If 1B does not appear at the top of your list, it could be due to the sorting order of your list. Make sure to check the PTW Templates and Sheet Sets

folders as well. To change the order, simply click on the Name column.

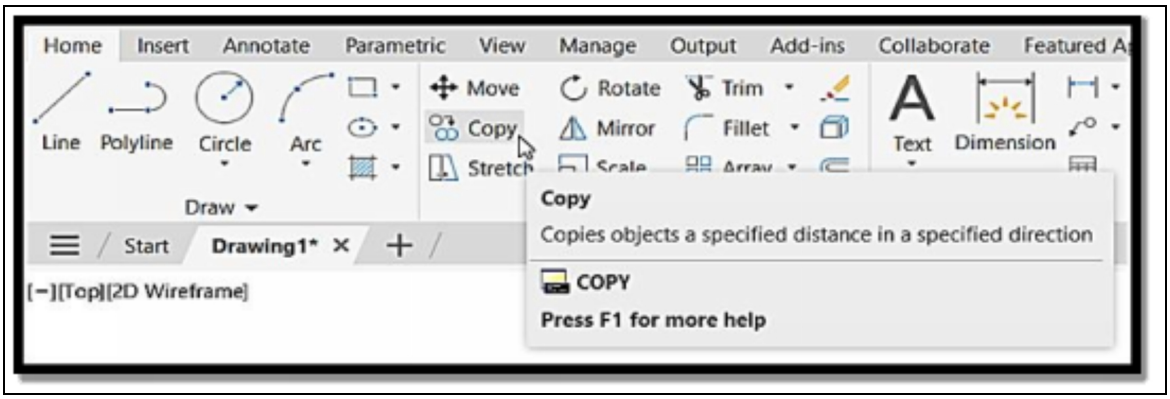
- To close the drawing window, simply click on the Close button (X) located in the upper-right corner. Ensure that you close the current drawing without closing the entire AutoCAD application window.
- To initiate a new project, simply select the **New tool** located on the Quick Access toolbar. Opening the Select template dialog box is a standard procedure. Now, 1B has been moved to the top of your list.
- Select **1B** or the file name you have used from the list of templates.
- Press the **Enter key** or click on the **Open button**. A new drawing is automatically generated with all the settings from 1B already preconfigured. Now, you have the flexibility to use the 1B template whenever you require a drawing that adheres to B-size limits while retaining your customized layers and settings.
- Enable or disable the grid according to your preference.

# How to Use the Copy Command

The COPY command functions similarly to the MOVE command. To begin, multiple copies of the circle are created and strategically placed within the drawing area.

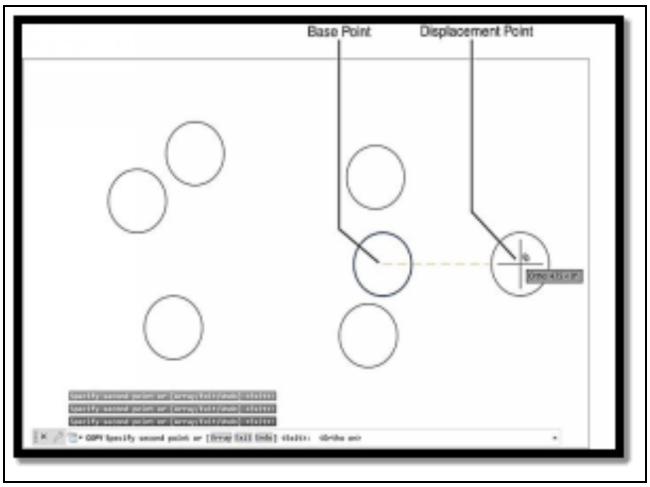
**Here's a helpful suggestion:**

- To access the Copy tool, navigate to the Modify panel located on the Home tab of the ribbon. Provide the coordinates for the base point. Indicate the location of the second point. Provide the coordinates for the second point or press <**Enter**> to exit the command.
- To access the Copy tool, navigate to the **Modify panel** on the Home tab of the ribbon. It's important to keep in mind that the letter "c" in AutoCAD is not used as an alias for the COPY command, but rather as an alias for the CIRCLE command.




**Use the copy tool to duplicate elements in your design.**

- Choose the circle.
- Use the right-click function to complete the selection of objects. When using AutoCAD, you will be prompted to specify a base point or displacement.
- Indicate the desired base point. Similar to the MOVE command, AutoCAD will request input for a second point.
- Provide the coordinates for the second point. A duplicate of the circle will be visible. The command line will display a prompt asking for the second point of displacement. At the end of the rubber band, you will notice another circle, accompanied by a Copy badge that resembles the COPY tool. AutoCAD is currently awaiting the addition of another vector, utilizing the identical base point as previously specified.
- Provide the coordinates for the second point. Feel free to repeat this process as many times as you desire. Once you delve into this field, you'll experience the exhilaration of effortlessly conjuring rings out of thin air and effortlessly placing them across the drawing area. Once the task is completed, your drawing should include multiple copies of the circle.

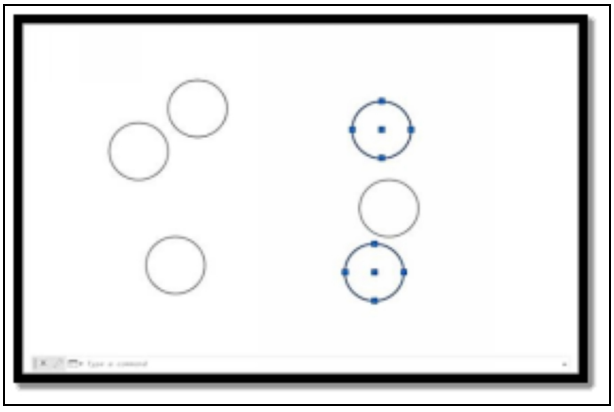



- Press the Enter key to exit the command.

# Copying with Grips

The grip editing system offers a unique method for generating multiple copies in all five modes. The functionality of the Copy option varies based on the Grip Edit mode. Currently, you can utilize the Copy option in the Move mode, which offers a convenient shortcut for duplicating the same process you recently performed using the **COPY command**. Since you have multiple circles in your drawing, now is the perfect time to master the technique of copying or moving multiple objects simultaneously using grip editing.

- Choose two circles from the options provided.

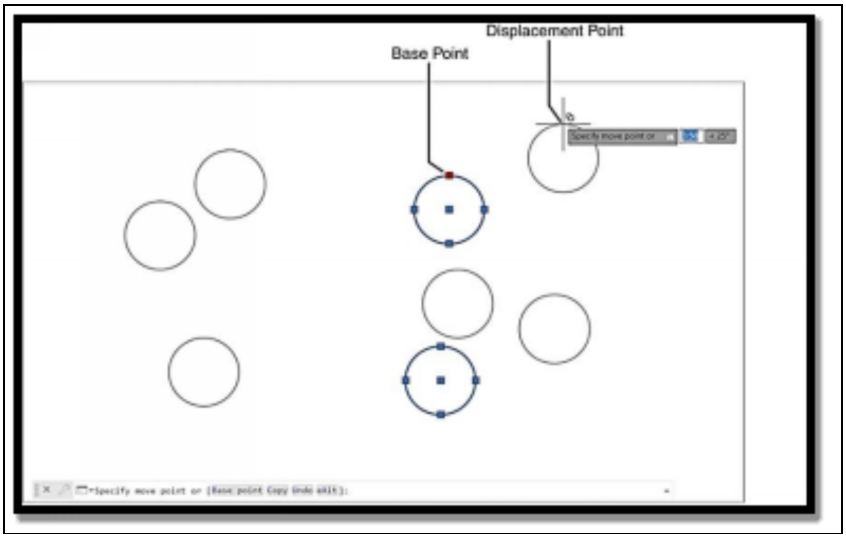


- Select a grip on one of the two circles that are highlighted. The color of the grip needs to be modified.

- To move the object, you can simply right-click and choose the **Move option** from the shortcut menu. You will now be directed to the Move mode prompt.

**Indicate the desired move point or choose from the available options: Basepoint, Copy, Undo, or Exit.**

- To begin copying, simply right-click and choose the **Copy option**. AutoCAD will request the move point once more. Copying in the grip editing system is always done in multiples. After entering this mode, you can effortlessly generate duplicates at any desired point until you choose to exit by typing 'x' or pressing the spacebar.
- Move the cursor to carefully observe the two circles that have been dragged.
- Indicate a point to duplicate the two highlighted circles.
- Provide an additional point to generate two additional copies.
- To exit the grip editing system, simply press <**Enter**> or the spacebar.
- Use the <**Esc**> key to easily remove grips.




# How to Use the MOVE Command

Being able to effortlessly move and duplicate objects is a major benefit of working with a CAD system. CAD is like the digital equivalent of drafting, just as word processing is to typing. Nowhere is this comparison more

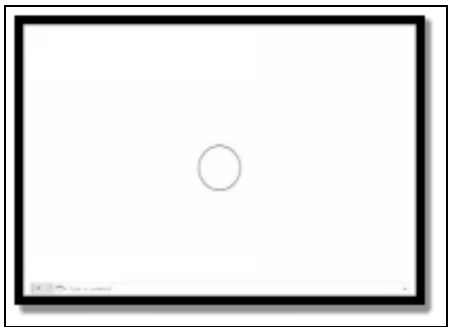
fitting than in the cut-and-paste capabilities provided by the MOVE and COPY commands.

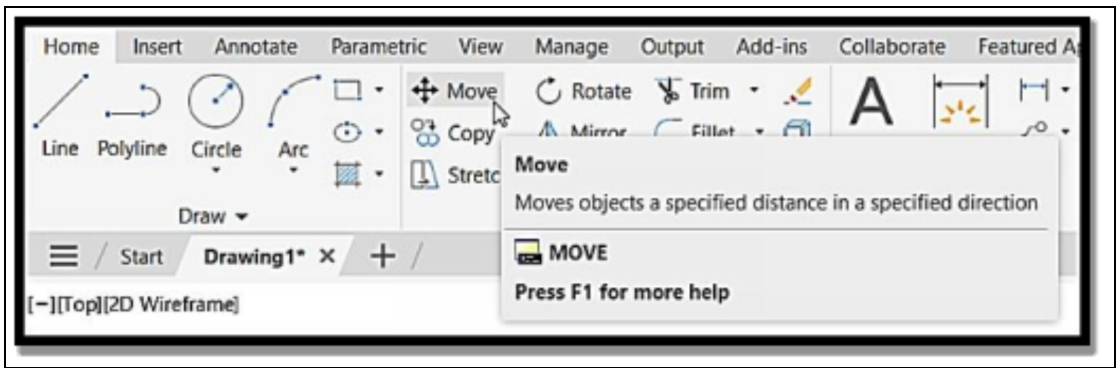
**Here's a helpful tip: Here is a step-by-step guide on how to use the MOVE command in AutoCAD:**

- Select the Move tool located in the Modify panel on the Home tab of the ribbon.

A selection set is a group of objects that have been chosen in AutoCAD for further editing or manipulation. (If noun/verb selection is enabled, you have the option to reverse the order of Steps 1 and 2.) Indicate the desired reference point for displacement. Provide the coordinates for the second point.

- Create a circle with a radius of 1 and position it near the center of the drawing area at coordinates (9,6).
- Select the **Move tool** located in the Modify panel on the Home tab of the ribbon. You will be prompted to select the objects that you want to move.



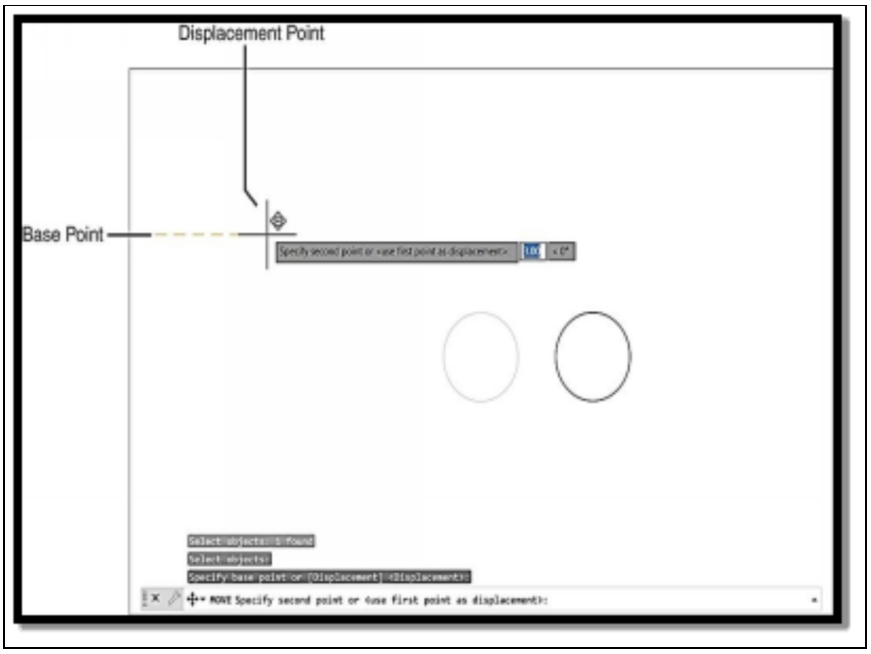


- Choose the circle. The circle you have selected is currently displayed with a blue highlight. AutoCAD provides real-time feedback in the command line, displaying the number of selected objects and encouraging you to continue selecting more.
- To conclude object selection, simply perform a right-click.

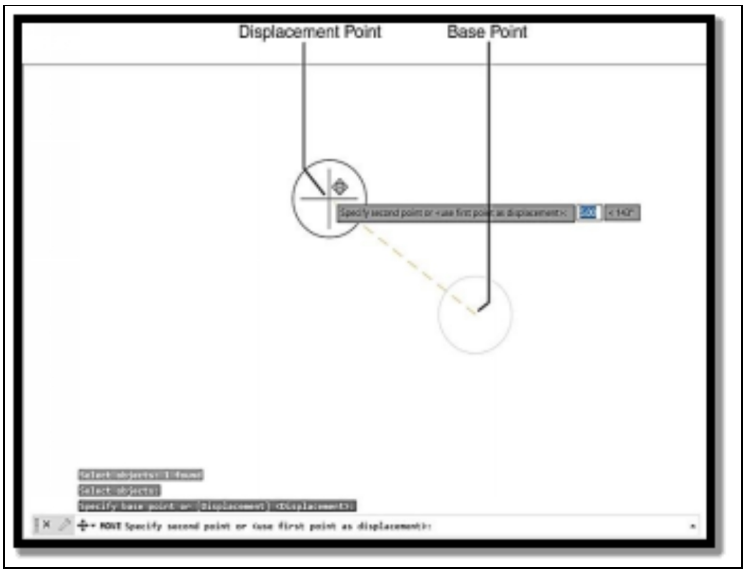
**Indicate the base point or provide the displacement value:** Typically, the movement is defined by specifying two points that indicate the desired distance and direction for moving the objects. You have the flexibility to choose the base point for moving objects, regardless of their location. Any point can be utilized to precisely define the desired movement of your objects. Initially, this may appear unfamiliar, but it will quickly become second nature. Certainly, it is possible to designate a specific point on an object if desired. When working with a circle, it can be quite convenient to utilize the center point.

- Enable **Snap Mode** if it is currently disabled.
- Choose a point that is positioned away from the left edge of the drawing area. When using AutoCAD, a rubber band will appear from the point you've selected, and the software will then prompt you to specify a second point.

When the cursor is moved in AutoCAD, a circle is automatically drawn to provide a visual representation of the movement. If you wish to shift the circle 3.00 units to the right, here's what you need to do.




- Observe the dynamic input tooltip and extend the rubber band until the tooltip displays 3.00<0°.
- Indicate a point that is 3.00 units to the right of your base point. After moving the circle, the rubber band and the original circle vanish, leaving you with a perfectly positioned circle. Now, attempt a diagonal movement.
- Disable **Ortho Mode** if it is currently enabled.
- To access the Move tool, navigate to the Modify panel on the Home tab of the ribbon. Alternatively, you can quickly repeat the command by pressing the spacebar. After executing the command, AutoCAD will display the "**Select objects**" prompt.
- Choose the circle.
- Press the <**Enter**> key or the spacebar to complete the selection of objects. AutoCAD will request the selection of a base point.
- Indicate the desired base point.
- Adjust the position of the circle by moving it diagonally in any desired direction.
- Reposition the circle closer to the center of the drawing area. Consider selecting the center point of the circle as the base point and utilize the coordinate display to pinpoint a point around (9,6) for your second point.




# Moving with Grips

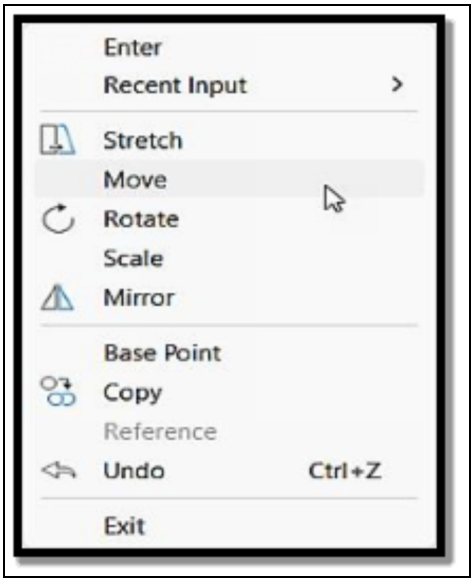
Use grips to execute a wide range of editing procedures. This editing method is straightforward to use. There are a few limitations to be aware of. When working in AutoCAD, you have the option to select objects using different methods such as pointing, windowing, or crossing.

- Choose the circle. The circle has been selected and grips are now visible. It is important to note that the grips for a circle are strategically positioned at the quadrants and the center. When it comes to more complex editing procedures, it's crucial to carefully consider which grip or grips to use. For this exercise, any of the grips will work perfectly.
- Gradually glide the cursor across one of the grips. When performing this task with precision, it becomes apparent that the cursor seamlessly attaches to the grip as it glides over it. When the cursor is locked onto the grip, the grip will change its color to orange. When you're positioned on any of the quadrant grips, you'll notice a value of 1.00 displayed, which represents the circle's radius.
- Click when the cursor is locked onto a grip.

The color of the selected grip undergoes another change, transitioning from orange to red. Stretching is one of the primary editing modes that can be accessed by selecting grips on objects. Stretch is a term in AutoCAD that

encompasses various meanings, often defying conventional expectations. This guide does not include information on the stretch editing mode or the STRETCH commands. Currently, it is recommended to disregard the stretch function and instead use the Move mode. AutoCAD offers a handy shortcut menu specifically designed for grip editing purposes.

- Perform a right-click action within the drawing area.




- Choose the **Move option** from the shortcut menu. The shortcut menu vanishes, and you find yourself in Move mode. A circle is added to the cursor for preview.
- Indicate a point within the drawing area. The circle will be positioned according to your specifications.
- Use the <**Esc**> key to easily remove grips.

# Moving by Typing a Displacement

Another method for using the MOVE command is available. Instead of specifying a distance and direction, you have the option to input a horizontal and vertical displacement. To move the circle 3 units to the right and 2 units up, follow this procedure (there is no automatic editing equivalent for this)

- Choose the circle.

- To initiate the Move tool, press the letter "**m**" on your keyboard and hit Enter. Alternatively, you can also click on the Move tool icon.
- When prompted for a base point or displacement, simply enter the values 3 and 2, followed by pressing the **Enter key**.
- Simply press the <**Enter**> key when prompted for the second point.

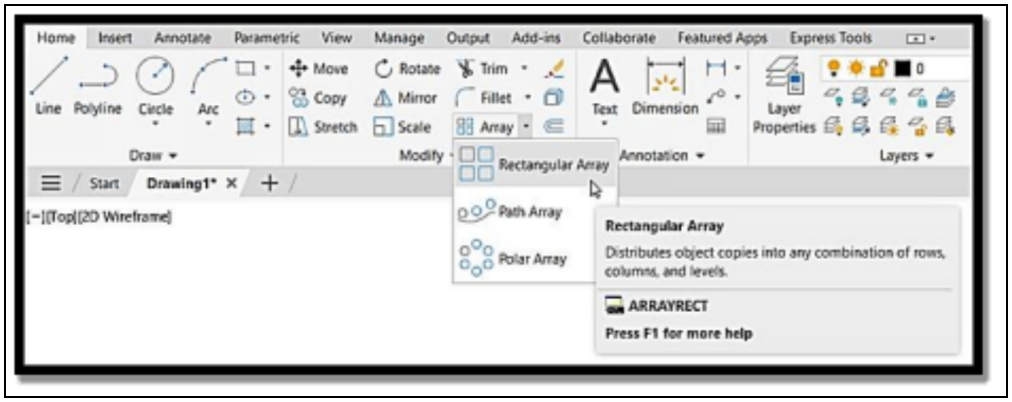
# How to Use the ARRAYRECT Command

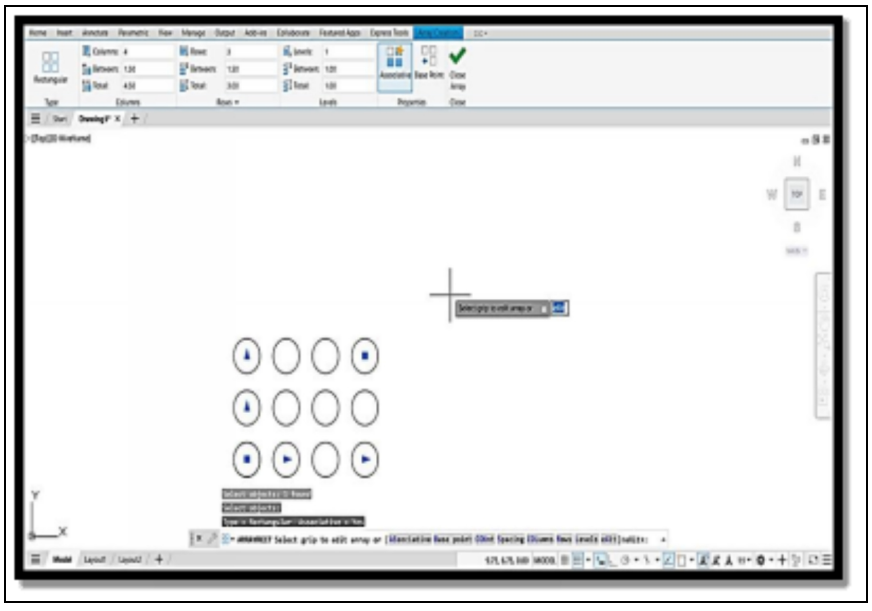
The ARRAYRECT command provides a robust alternative to basic copying, offering enhanced functionality. An array is a way to represent an object by repeating it in a matrix format. The command allows for the duplication of objects or groups of objects, placing them in precise, evenly spaced positions based on mathematical calculations. Once defined, an array becomes a versatile template that can be modified in various ways. An array is a pattern of objects that can be rectangular or circular, or even follow a specific path. There are three different types of arrays in AutoCAD. Rectangular arrays are organized linearly, with clear rows and columns to define their structure.

Polar arrays involve the angular distribution of objects around the circumference of an arc or circle. Path arrays can be easily generated by duplicating objects at regular intervals along any curve, line, or closed figure. The grid lines demonstrate a rectangular array, while the radial lines on a circular dial showcase a polar array. Path arrays offer a wide range of possibilities in terms of shapes and forms. In this chapter, we will delve into the intricacies of rectangular arrays. Rectangular arrays are determined by specifying the number of rows, the number of columns, and the spacing between each element. To begin this exercise, remove all the circles from your drawing. It's a great chance to explore the All selection option in the **ERASE command**.

- Select the **Erase tool** located in the Modify panel on the Home tab of the ribbon.
- Enter all <**Enter**>. All circles will be displayed in a gray color.
- Press the Enter key once more to finalize the command.

- Let's create a circle with a radius of 0.5, perfectly centered at the coordinates (2,2).
- Select the **Rectangular Array tool** located in the Array drop-down menu of the Modify panel on the Home tab of the ribbon. To access the Rectangular Array tool, navigate to the Modify panel and locate the menu where the Polar Array or Path Array tool is displayed. From there, simply click on the **Rectangular Array tool**. To execute the ARRAYRECT command, you will be prompted to select the objects.
- Choose a circle with a radius of 0.5.
- To conclude object selection, simply perform a right-click.



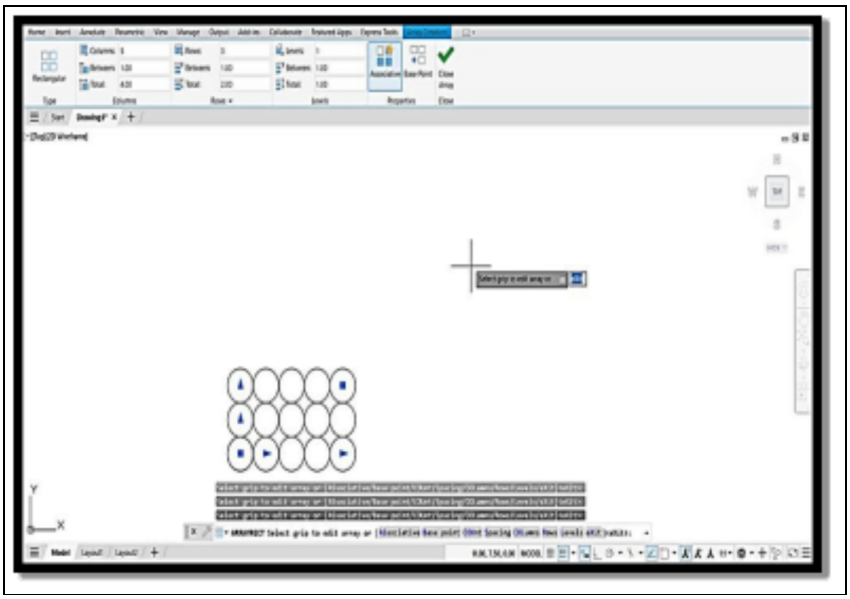
# The Array Creation Contextual Tab

Under specific conditions, AutoCAD will open a ribbon tab that is not typically accessible. This tab is referred to as a contextual tab. This is the initial illustration of a contextual tab, which becomes visible once the ARRAYRECT command is initiated. When examining this tab, you will notice the Type designation (Rectangular) and the four panels: Columns, Rows, Levels, and Properties, arranged from left to right. There are three edit boxes for entering specifications. When your cursor is placed on any of these boxes, a tooltip box will appear, providing the specification details. Based on the given information, the array consists of 4 columns, 3 rows, and 1 level. The count is represented at the top. Levels are an essential aspect of arrays, providing the third dimension. In the context of 2D arrays, levels will consistently have a count of 1. The spacing between columns can be adjusted using the second edit box in the Columns panel. From the preview array, it is evident that the column spacing is set at 1.50 units. In the Rows panel, the spacing between rows is set to 1.50. The level spacing is set at 1.00, although it does not hold any significance in your two-dimensional array. The third value represents the overall distance occupied by the array. The measurements are 4.50 units horizontally across the columns and 3.00 units vertically along the rows. A contextual tab is a ribbon tab that will open automatically when a specific condition is met. When a previously drawn array is selected, the Array Creation tab will open. To modify these values and generate an array according to your specific requirements, you can easily update the values in the edit boxes. In this exercise, we will create a 5 × 3 array with 1.0 unit spacing between rows and columns. To accomplish this, you can modify the values in the contextual tab as demonstrated.

# Note

If you happen to exit the ARRAYRECT command, you can easily reopen the contextual tab by selecting any item within the array. When you select an array, the Array contextual tab will be displayed instead of the Array Creation contextual tab. The options on both tabs are identical, although it's worth noting that the names may vary from the instructions provided in the exercise.

- In the Array Creation contextual tab, simply click on the **Columns edit box** and enter the value of 5, then press **Enter**. Pressing <**Enter**> will automatically add a column, resulting in a total of 5 columns.



# Important

**The symbol located on the right side of the edit box signifies the ability to generate arrays using mathematical relationships, expressions, and absolute values.**

- Enter the value of 1 in the edit box located between the columns panel. The column spacing will be decreased to 1.00.
- Enter the value '1' in the edit box located between the Rows panel. Press the **Enter** key to confirm.
- To exit the ARRAYRECT command, simply press the Enter key. Alternatively, if you have selected the array for editing and want to clear the grips, press the Esc key.

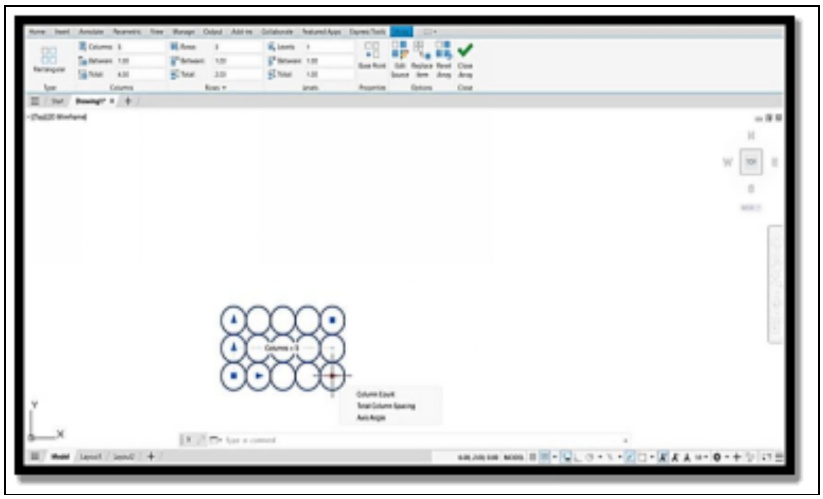
# Using Multifunctional Grips

One way to enhance the flexibility of array specifications is by using multifunctional grips. When the array is selected, six grips will be displayed. These grips can be utilized to modify the values that define the

array. Every grip in AutoCAD serves a distinct function and is restricted to modifying specific values.

- Choose the array.
- Position the cursor over the triangular grip located at the lower-right corner of the array. When the cursor is positioned on the grip, a tooltip will appear displaying three available options. By choosing from these options, you can utilize this grip to modify the array in three distinct manners. This grip is known as a multifunctional grip. When the tooltip is displayed, you can hover the cursor and choose from the available three options. The first option, Column Count, in this case, is the default choice. With Column Count, you can effortlessly incorporate additional columns into the array while keeping the spacing and row count intact. With the Total Column Spacing option, you can adjust the spacing between columns while keeping the number of columns unchanged. With the Axis Angle option, you can tilt the columns at an angle.

A multifunctional grip is a versatile tool that allows for editing objects in various ways. When a multifunctional grip is selected, AutoCAD will present a list of available editing modes for you to choose from.



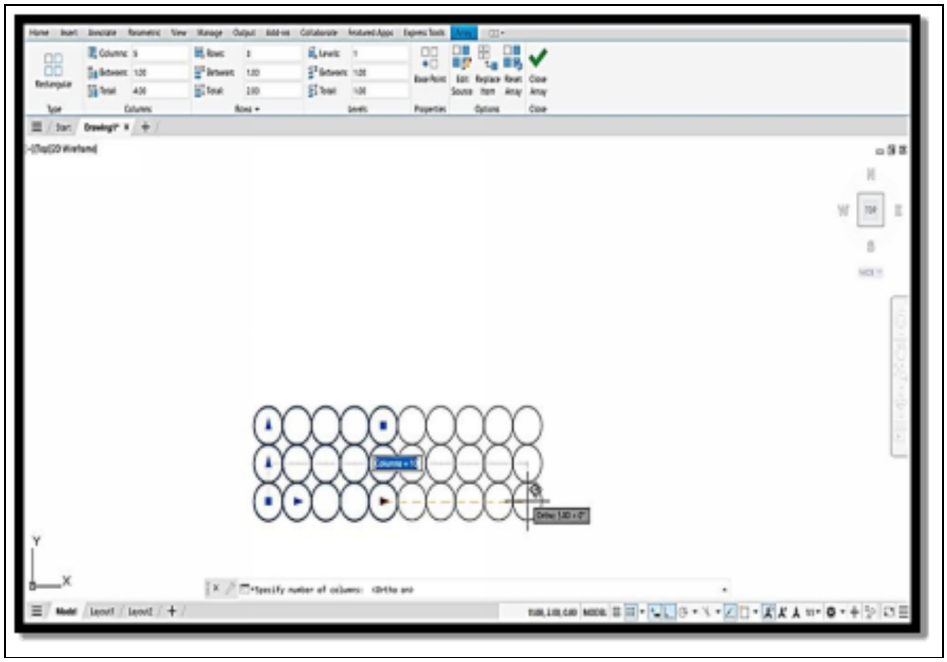
By simply hovering the cursor over different grips, you'll discover that each grip has its own unique set of functions. There are two grips located above and to the right of the lower-left corner, which are the only exceptions. These options are specifically designed to modify the spacing between rows

and columns. As a result, these two functions lack versatility and do not provide a menu of options.

**To add five additional columns, simply use the lower-right corner grip.**

- Locate and select the triangular grip situated at the lower-right corner. A tooltip will be displayed, indicating the total number of columns in the array. As you navigate the cursor towards the right, a preview of the columns being added in that direction will appear.
- Adjust the cursor towards the right until you have included five more columns, resulting in a total of ten.

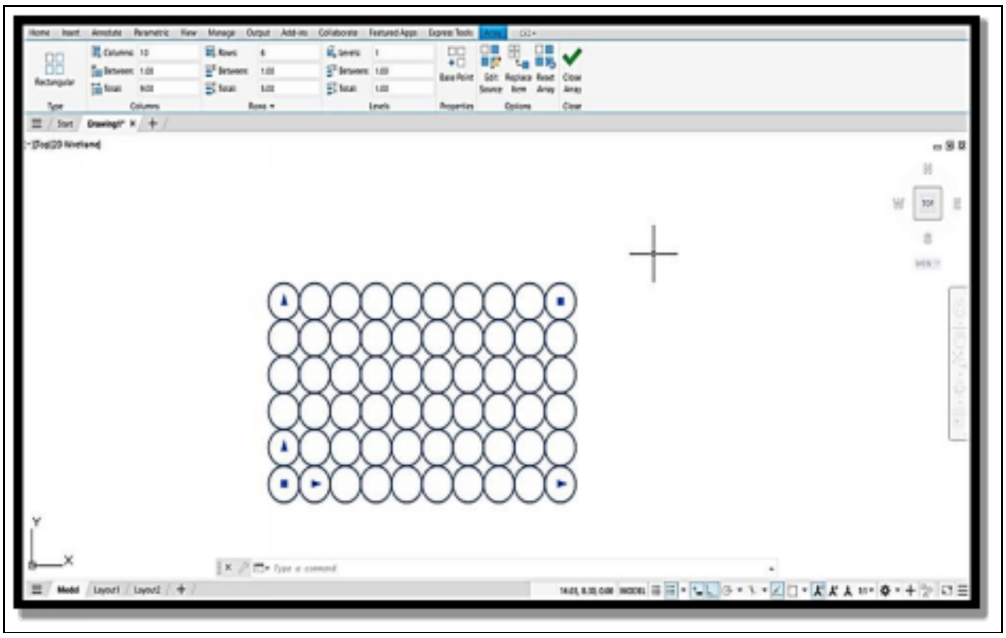
Observe the elongated preview grid and the accompanying message, indicating that there are 10 columns.



**Columns have been added for previewing, along with a comprehensive list of multifunction grips.**

- Simply click in the drawing area to finalize the modification. It is important to observe that the grips on the right side extend towards the right side of the array. Use the top-left grip to effortlessly incorporate three additional rows to the top of the array.

- Locate and select the triangular grip positioned at the top-left corner of the array.
- To add three additional rows, simply move the cursor upward.



- Take a moment to analyze the alterations in the contextual ribbon tab.

# How to Adjust Plot Settings

Previewing plots can be a valuable time-saving technique that also reduces paper waste and helps you learn more efficiently. Before diving into this exploration, it's important to have a drawing or objects already sketched in your design. This will allow you to closely observe the impact of different modifications you make. Ensure that the drawing you are currently working on utilizes the 1B template, which will automatically set the limits to 18 × 12. The circles created in the previous exercise will be suitable for this task. Access the Plot tool by clicking on it in the Quick Access toolbar.

# The Printer/Plotter Panel

Choosing the right plotter is a fundamental aspect of configuring a plot in AutoCAD. Various plotters use a variety of sheet sizes and come with their unique default settings. Start by reviewing the list of available plotting devices and then proceed to include a plotter in the list.

- To access the Name drop-down list in the Printer/plotter panel, simply click on the arrow located to the right. The list displayed is determined by your system and may consist of printers, plotters, and any other electronic output devices that have been configured. Additionally, you will find the AutoCAD PDF (General Documentation).pc3 option, which allows you to conveniently send a drawing via email or share it online.

# Add-a-Plotter Wizard

To fully delve into AutoCAD plotting, it is crucial to have a plotter at your disposal. Make sure to select one from the Printer/plotter list. If you only have a printer, your options for drawing sheets may be somewhat limited. There might be only one option available, such as an A-size. Within this chapter, users have the option to utilize the AutoCAD PDF (General Documentation) device to replicate a plotter, or they can employ the Add-a-Plotter Wizard to install one of the standard plotter drivers provided by AutoCAD. This can be done even if there is no physical plotter present in the system.

**Here is the procedure to add a plotter:**

- Close the Plot dialog box.
- To access the Plotter Manager, navigate to the ribbon and click on Output, followed by Plot, and then **Plotter Manager**.
- To access the Add-a-Plotter Wizard, simply double-click on the shortcut located in the Plotters window.
- Proceed to the next step by clicking on the "**Next**" button located on the Introduction page.
- Ensure that My Computer is selected on the Add Plotter – Begin page, and proceed by clicking Next.
- Choose a manufacturer and model from the options on the Plotter Model page, and then proceed by clicking **Next**.
- Proceed to the next step by clicking on the "**Next**" button located on the Ports page.
- Proceed to the next step by clicking on the Next button located on the Plotter Name page.
- Select the "**Finish**" option located on the Finish page. Once the wizard has finished, the newly created plotting device will be

accessible from the list of plotting devices in the Plot dialog box.

- To redisplay the Plot dialog box after installing a plotter driver, locate and select the Name drop-down list situated in the Printer/plotter panel.
- Choose a plotter or the AutoCAD PDF (General Documentation).pc3 from the Name drop-down list to simulate a plotter.

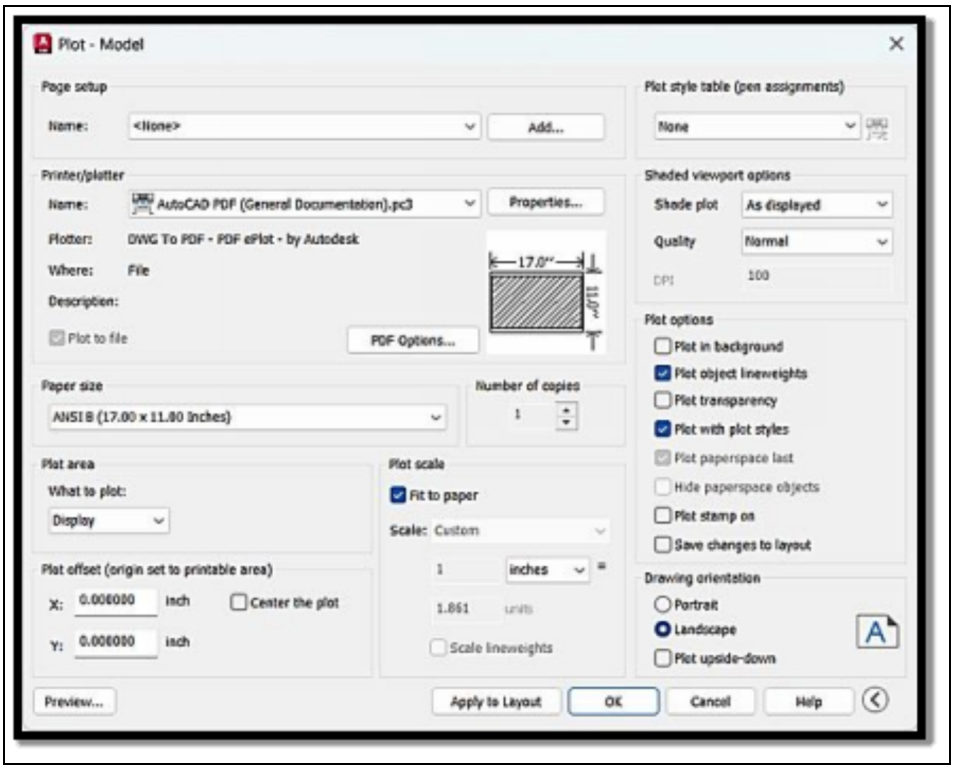
# Paper Size

**Once you have chosen a plotter, you will be presented with various paper-size options.**

- Choose a B-size drawing sheet from the drop-down list in the Paper size panel. The size will vary based on the plotter you have chosen. A commonly chosen sheet size is ANSI B 17 × 11.

# Understanding Drawing Orientation

**The options for selecting drawing orientation can be found in the expanded Plot dialog box.**




- To expand the dialog box, simply click the right arrow button (>) located at the bottom right corner. There are two main options for drawing orientation: portrait and landscape. In portrait orientation, the short edge of the paper is across the bottom, while in landscape orientation, the long edge is across the bottom. A portrait orientation is commonly used for letter or printed sheets of text, while a landscape orientation is typically used for drawing sheets. By using the radio buttons, you have the flexibility to print in any orientation, regardless of the default setting of your plotting device. It is important to consider the drawing orientation as it significantly affects the utilization of the plotting area on the page. Therefore, it is recommended to always review the partial preview whenever you change orientations. Here's what you can do:
- To view the interpretation of your current drawing orientation, simply click on the **Preview button**.
- To return to the Plot dialog box, simply press the <**Esc**> or <**Enter**> key.
- To change the orientation setting, simply switch between Landscape and Portrait.

- To observe the plot's alteration based on drawing orientation, click the **Preview button** once more.

**Here's a helpful tip:** Certain plotters offer a variety of paper orientations for your selection. As an AutoCAD expert, you may come across different options for paper sizes, such as 11 × 17 and 17 × 11. When selecting a paper for your drawing, it is important to consider the correlation between the paper and the drawing orientation. To plot in the landscape, choose the 17 × 11 option. For portrait, select the 11 × 17 option. Otherwise, if your paper settings are not aligned with your drawing orientation, it can lead to confusion.

- To complete the exercise, adjust the drawing orientation to Landscape. This is the standard setting for the majority of plotters.

# Plot Area

Take a look at the Plot area panel located on the left side of the Plot dialog box. This is a vital component of the dialog box that enables you to precisely define the section of your drawing that will be plotted. You have several options to choose from **Window, Display, Limits**, and **Extents**. Modifications made in this area have a substantial influence on the optimal plotting space. The drop-down menu displays the available choices for the plotting area. The display function generates a plot based on the contents of the drawing area. When utilizing the ZOOM command to magnify a specific section of the drawing before selecting the PLOT option, AutoCAD will plot whatever is currently displayed in the drawing area. As you are aware, limits are set using the LIMITS command in AutoCAD. When utilizing the standard Architectural B-size template and choosing Limits, the plot area will be set to 18 × 12. The term "extents" is used to describe the specific area within a drawing where objects have been created. The size can vary beyond the drawing's boundaries. Experiment with different selection options such as Limits, Extents, Display, and Window. Don't forget to click the Preview button to observe the outcomes. When making changes, it's important to keep an eye on the Plot scale boxes, which indicate the conversion between inches and units. If the "**Fit to paper**"

option is selected, you will observe notable modifications in the scale ratios as AutoCAD automatically adjusts the scales based on the specified area.

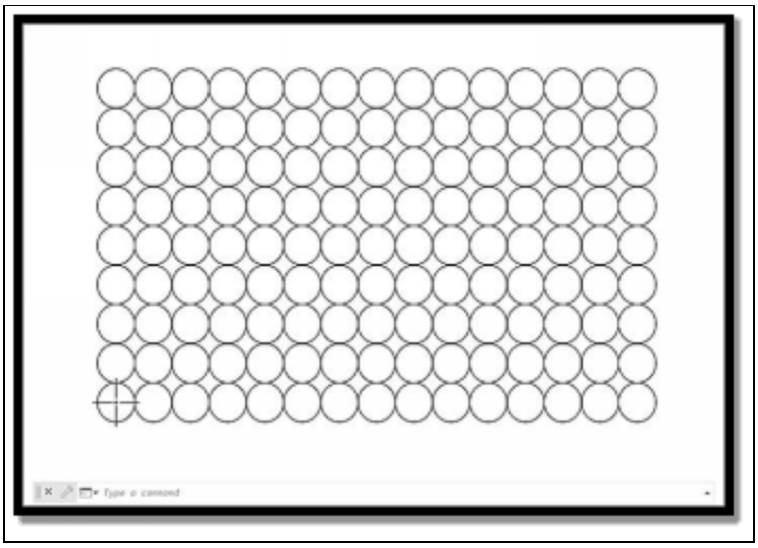
# Plot Offset

The Plot offset panel can be found in the lower-left corner of the Plot dialog box. The plot offset feature allows you to precisely position the plot area on the drawing sheet. More precisely, it determines the placement of the plot's origin. By default, the origin point (0,0) is positioned at the lower left of the plotted area, serving as the reference for other locations. When a different offset specification is entered, such as (2,3), the origin point of the drawing area is positioned accordingly. From this new point, plot locations are determined. The placement of objects on paper is significantly impacted by this. Another option in Plot offset is to select the Center plot setting. AutoCAD automatically aligns the drawing so that the center point of the plot area matches the center point of the drawing sheet. Experiment with different plot offset combinations, such as centering the plot, and use the Preview button to view the outcomes.

# How to Create Center Marks

Here, you'll find a brief overview of a straightforward dimensioning feature that is commonly used towards the conclusion of the chapter. Finally, the exercise is concluded with a thorough review of the procedures for creating rectangular arrays that were previously introduced.

- Before we proceed, it is important to have the array of circles from the previous section in your drawing. With the CENTERMARK command, you can easily add a center mark with centerlines to a circle.

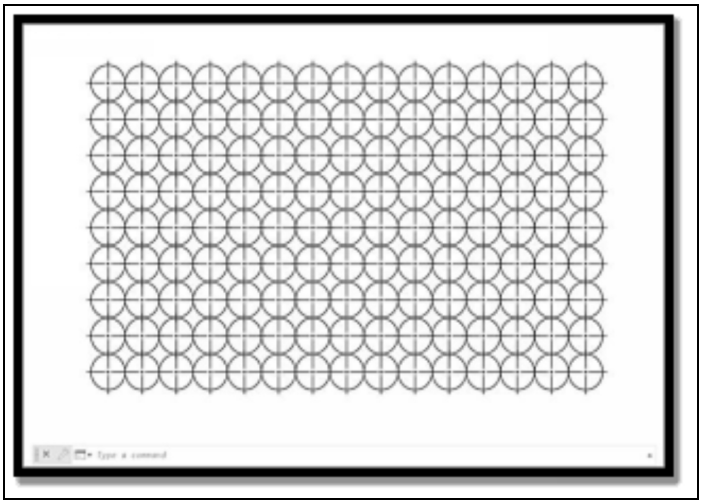


- Press the "**Enter**" key. As you type, the autocomplete feature will display a list of commands that start with "ce". Choose **CENTERMARK** from the available options. AutoCAD provides prompts:

**Choose either a circle or an arc to add a center mark.**

- Choose the circle located at the bottom-left corner of the array. AutoCAD automatically includes a center mark and centerlines, as depicted in the illustration. This process is extremely straightforward. For completion, the center marks are arranged in the same matrix as the circles in the drawing area. To add a center mark to each circle, follow these steps. To achieve this, simply create a rectangular array using the identical specifications as the circle array.
- Press the **Enter key** to execute the command.
- To access the Rectangular Array tool, navigate to the Modify panel on the Home tab of the ribbon and select it from the Array drop-down menu. When using ARRAY in AutoCAD, you will be prompted to select the objects you want to work with.
- Press the "L" key and hit Enter to select the most recently drawn object, which in this case is the center mark.
- To end object selection, you can either right-click or press the **Enter key**.

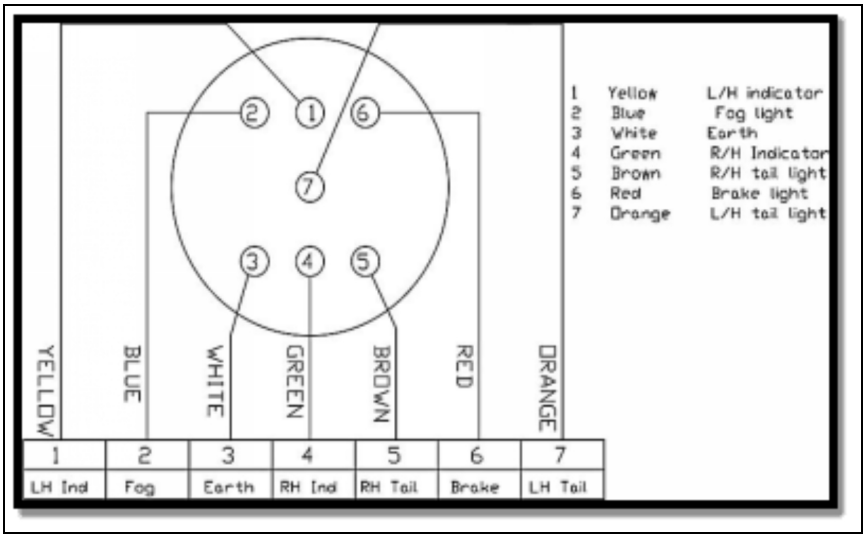
- Within the Columns panel of the Array Creation contextual tab, you can easily specify the desired number of columns, which in this case is 15. Additionally, you can set the precise distance between each column to be 1 unit.
- Within the Rows panel of the Array Creation contextual tab, specify the desired number of rows as 9 and the desired distance between rows as 1. Press the **Enter key** to execute the command.
- Use the <**Esc**> key to easily remove the grips.



# Chapter Drawing Projects

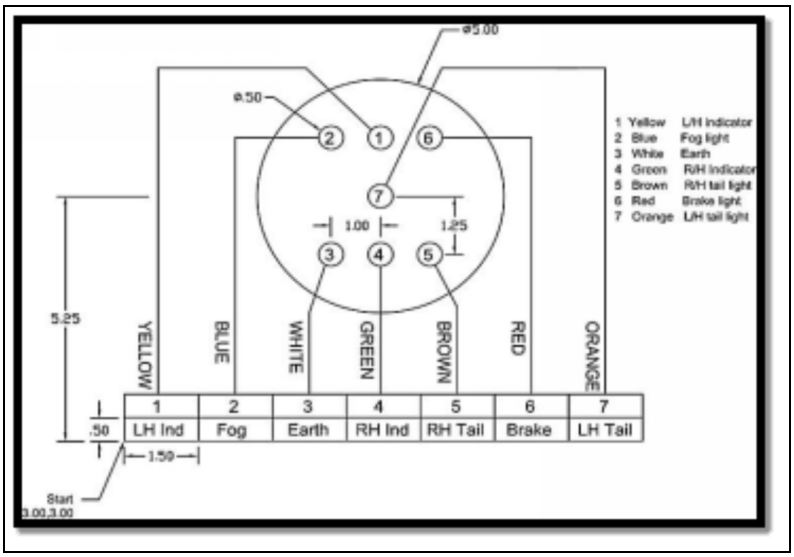
- **Drawing 4-1: *7-Pin Trailer Wiring Diagram* [BASIC]**

All the drawings in this chapter use the 1B template. It is important to note that there may be instances where you will need to modify settings. Layer settings remain consistent, while limits may vary periodically, and grid and snap settings are subject to frequent adjustments. This drawing provides an opportunity to enhance your skills in using the **COPY command**. There are multiple methods available for creating the drawing. Locations are only approximate unless specifically dimensioned. To ensure efficient organization, it is recommended to create a separate layer for each wire color. This will allow you to easily distinguish and manage the different components of your design. Once the layers are set up, you can proceed to draw the wire, box, circle, and text using the corresponding color layer. This approach will help maintain consistency and clarity throughout your AutoCAD project.



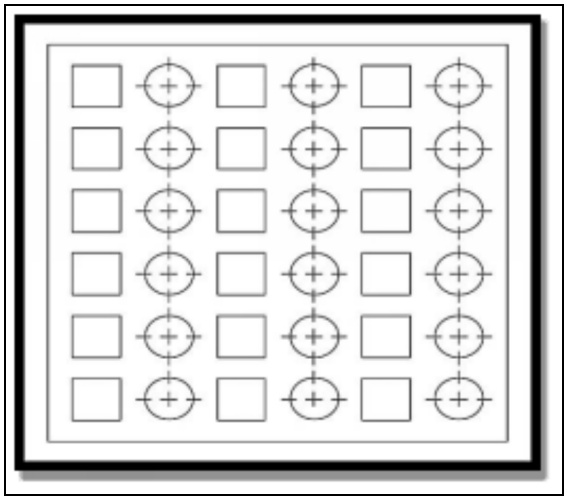

**Here are some suggestions for your drawings:** For the larger text, set the height to 0.25, and for the smaller text, set it to 0.18. Also, make sure the grid is set to 0.5 and the snap is set to 0.25. Create a rectangle with dimensions of 0.25 units by 1.50 units, starting at the coordinates (3.00, 3.00). To achieve a 2 × 7 array, you can use the array command to duplicate the rectangle. First, you'll want to create the circles using the appropriate tools in AutoCAD. Once you have the circles in place, it's time to connect

the wires. Make sure to use the appropriate commands to create accurate and precise connections between the circles. Pay attention to the details and ensure that the wires are properly aligned and connected. With your expertise in AutoCAD, you'll be able to effortlessly complete this task. Use the Nearest object snap to precisely align the wire lines with the circles. Place all text in their respective approximate positions.



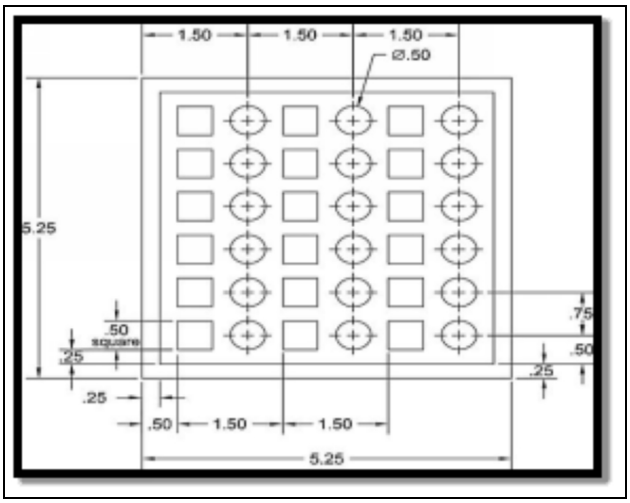

- **Drawing 4-2:** ***Grill*** **[BASIC]**

To expedite the process, you can use the **ARRAYRECT command** and adjust the **DIMCEN system** variable to 0.09.



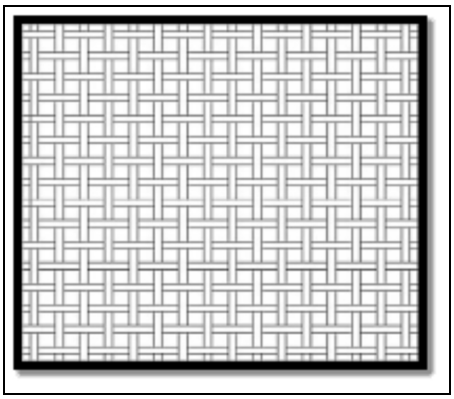
**Here are some suggestions for your drawings:** Set the grid spacing to 0.5 units and the snap spacing to 0.25 units. Start by creating a square with

dimensions of 5.25 × 5.25. Adjust the dimensions by 0.25 units uniformly to form the inner square. Start by drawing the small square and the circle with centerlines in the lower-left corner. Once you have those in place, use the ARRAYRECT command to easily create the remaining elements. To change the system variable to 0.09, simply type "**dimcen**" and press **Enter**. Next, enter the command "**dimcenter**" followed by pressing the **Enter key**. Then, type "**cen**" and press Enter to draw the centerlines on the circle. Before arraying the square and the circle, it is important to complete this step. Additionally, it's important to note that you can easily reverse a misplaced array by using the **U command**.

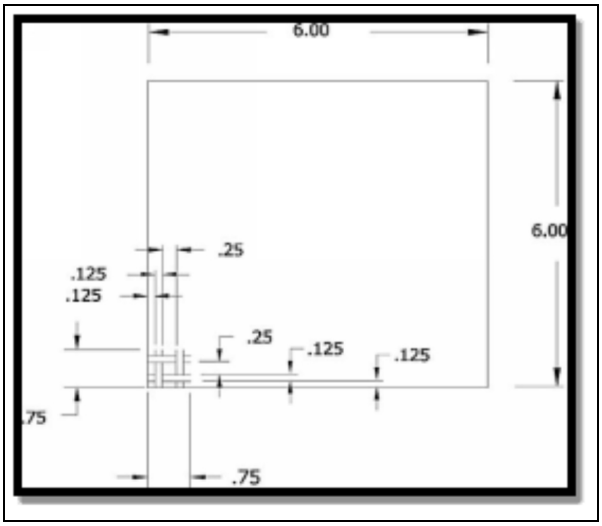



- **Drawing 4-3: *Weave* [INTERMEDIATE]**

As you create this drawing, marvel at the efficiency of AutoCAD and consider the significant time savings compared to manual drafting! The final drawing closely resembles the depicted representation. To provide clarity, the drawing displays a single cell of the array along with its dimensions.

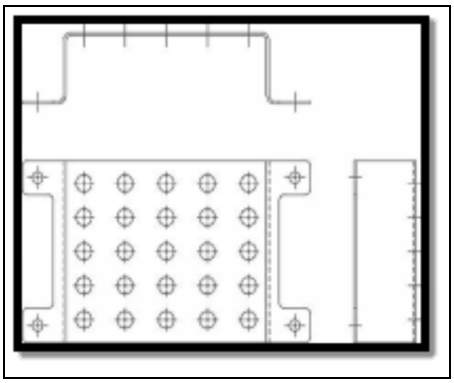


**Here are some suggestions for your drawings:** Set the grid spacing to 0.5 units and the snap spacing to 0.125 units. Create a 6 × 6 square and then zoom in on the lower-left area using the window command. This area is depicted in the lower left corner of the dimensioned drawing. Take careful note of the dimensions and accurately create the line patterns for the lower-left corner of the weave. If you prefer, you have the option to use the **COPY command** in various locations, although the time saved would be relatively insignificant. Do not be concerned if you need to make some adjustments to get this just right; once you have achieved the correct configuration, the remaining steps will be straightforward. To repeat the lower-left cell in an 8 × 8 matrix, you can use the **ARRAY command**. If you make an error, use the "U" command and attempt the task again.

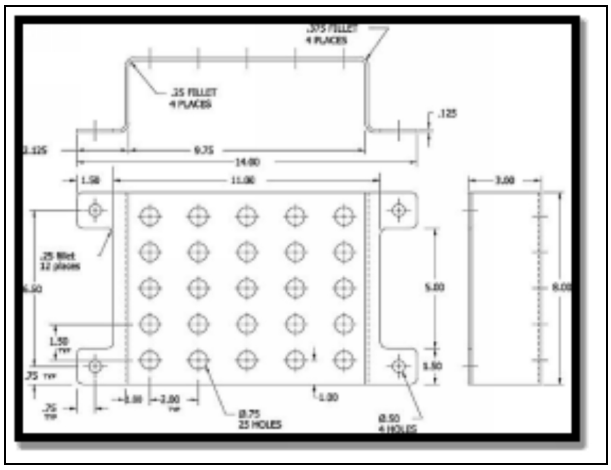



- **Drawing 4-4: Test Bracket [INTERMEDIATE]**

This drawing is an excellent opportunity to apply and reinforce the knowledge you have acquired so far. Take note of the recommended snap, grid, scale, and limit settings. Use the **ARRAYRECT command** in conjunction with the 0.09 centerline dimension to accurately draw the 25 circles on the front view.

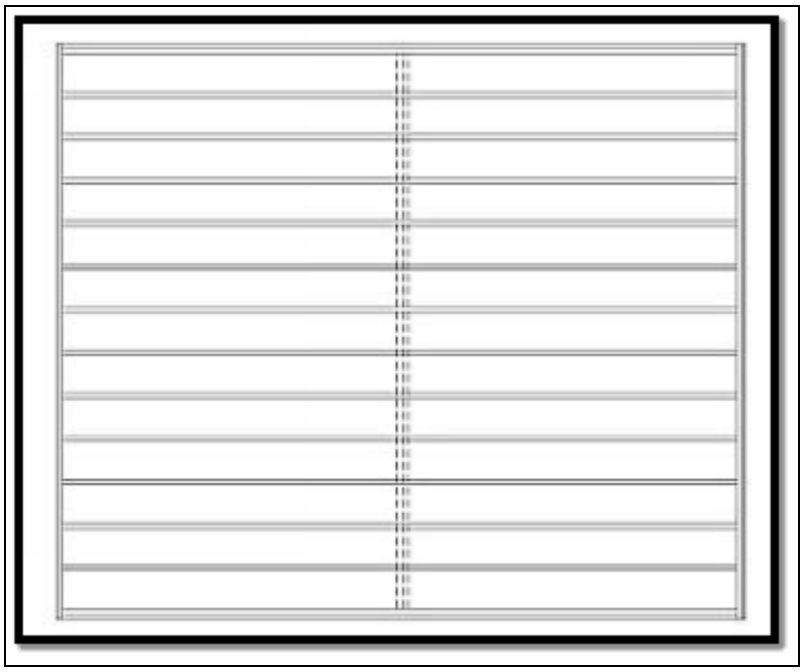


**Here are some suggestions for your drawings:** Set the grid spacing to 0.25 units, the snap spacing to 0.125 units, and the line type scale to 0.50. The drawing limits are defined as (0,0) to (24.18). Ensure that all lines are accurately placed on the appropriate layers, following their linetypes. It is important to draw centerlines through the circles before copying or arranging them. Failure to do so will result in the need to manually draw the centerlines on each circle or repeat the array process. Using the multiple copy function is a great way to efficiently create four holes with a diameter of 0.50. A rectangular array would be the ideal choice for accommodating the twenty-five holes with a diameter of 0.75. To ensure accuracy, it is recommended to adjust the **DIMCEN system** variable to 0.09 if you haven't already done so. Once you have drawn your initial circle, you can enhance it by using the "**dimcenter**" command. Simply type "**dimcenter**" followed by the Enter key, and then proceed to add centerlines to the circle. Generate the rectangular array of circles with centerlines as depicted.



- **Drawing 4-5: Floor Framing [ADVANCED]**

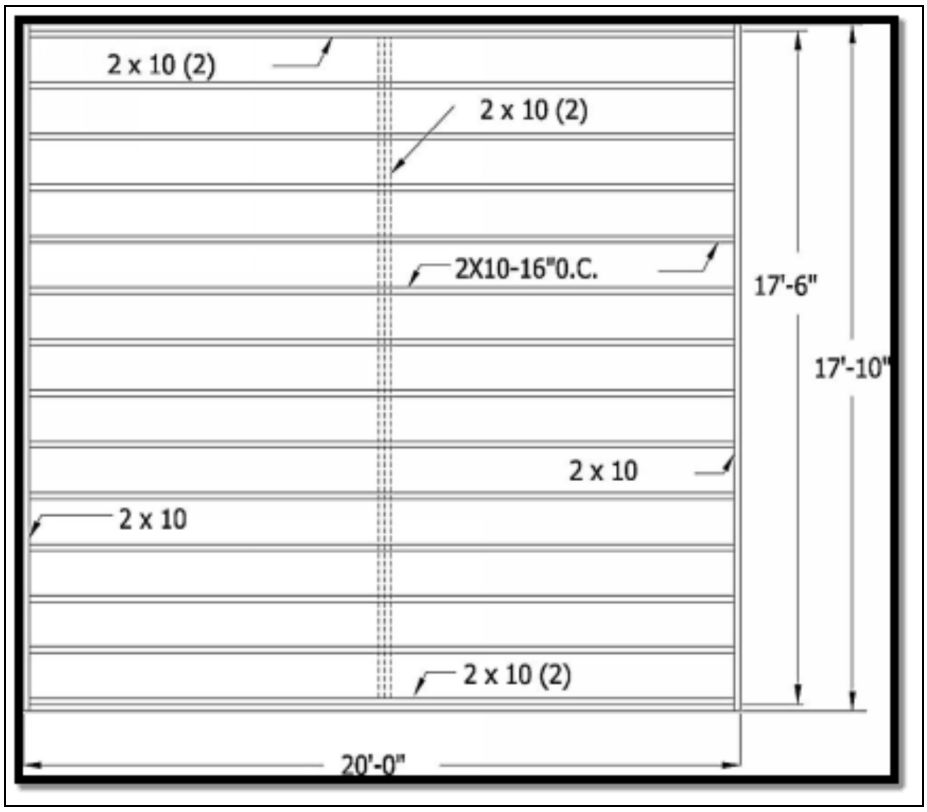
For this architectural drawing, several aspects of your drawing setup need to be modified. It is crucial to carefully consider the recommended settings.



**Here are some suggestions for your drawings:** Set the units to Architectural, with a precision of 0 feet and 00 inches. The drawing limits are set to 36 feet by 24 feet, with a grid spacing of 1 foot and a snap spacing of 2 inches. The line type scale is set to 12.

# Important

- It is important to note that lumber has dimensions that may differ from the nominal dimensions. As an AutoCAD expert, it's important to understand that a 2 × 10 measurement is equivalent to 1½" × 9¼". It is customary to utilize nominal dimensions in all drawings.
- Make sure to use the foot (') and inch (") symbols when establishing limits, grid, and snap settings, while excluding ltscale.
- Start by creating a rectangle with dimensions of 20' × 17'-10". Place the lower-left corner approximately at coordinates (4',4').
- To finish the left and right 2 × 10 joists, simply duplicate the vertical 17'-10" lines, positioning them 2" inward from both sides.
- Create a horizontal line that is 19'-8" in length, positioned 2" above the bottom. Duplicate this line 2" higher to complete the double joists.
- To create the desired array, you can use the Array command in AutoCAD. Specify the number of rows as 14 and the number of columns as 1. Set the spacing between rows to 16 inches. This will arrange the inner 2 × 10s in a neat and organized manner.
- Change the layer to 2 and proceed to draw the three hidden lines along the center.




OceanofPDF.com

# CHAPTER SEVEN
# UNDERSTANDING DIMENSIONS

Being able to dimension your drawings and incorporate crosshatch patterns significantly elevates the overall quality and functionality of your work. The dimensioning features in AutoCAD are a sophisticated system of commands, subcommands, and variables that can automatically measure objects and generate dimension text and extension lines. AutoCAD offers a range of dimensioning tools and variables that allow you to effortlessly create dimensions in various formats. These formats can be conveniently saved as styles for future use. One of the major benefits of CAD is the significant time saved by not having to manually draw each dimension object line by line.

## How to Create and Save a Dimension Style

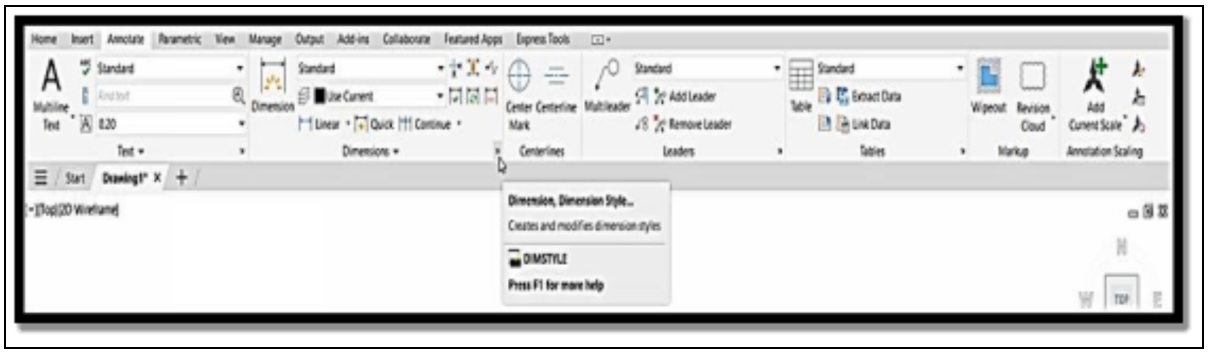
Dimensioning objects in AutoCAD is made much easier compared to manual dimensioning. Creating dimensions in AutoCAD is a fairly simple process. However, defining dimension styles to control the appearance of dimensions can be quite intricate. This complexity arises from the need to provide enough flexibility to accommodate various dimension styles. In the exercises that follow, you will gain a comprehensive understanding of managing dimension styles, using various options, and achieving desired results in AutoCAD dimensioning. Discover the process of creating a fundamental dimension style and applying it to accurately depict standard dimensions and tolerances. There are numerous potential variations for you to explore. When working in AutoCAD, it's crucial to start by assigning a name and defining a dimension style. A dimension style encompasses a collection of dimension variable settings that govern the appearance of text and geometry for all available dimension types. Creating new dimension styles in your drawing template and saving them can save you time and effort in the future. By doing so, you won't have to repeat these changes every time you begin a new drawing. **A dimension style is a collection of dimension variable settings that govern the appearance of text and geometry for all available dimension types.**

- To start, create a new drawing with limits set to 18 × 12. Alternatively, you can open the 1B template drawing if you have it. To achieve the desired formatting, adjust the dimension style settings accordingly. Ensure that dimensions displaying distances are set to show two decimal places, while angular dimensions are configured to have no decimals.

**Here's a helpful suggestion:**
When you modify a dimension style in a drawing template, those modifications will automatically apply as the default dimension settings for any new drawing that is based on that template. When locating drawing template files, they can be found in the Template folder. To access the Select File dialog box, simply use the **OPEN command**. Choose the **Drawing Template (*.dwt) option** from the drop-down list in the Files of Type section. The Template folder is opened automatically. Choose the desired template file from the provided list, then click on the **Open button**.

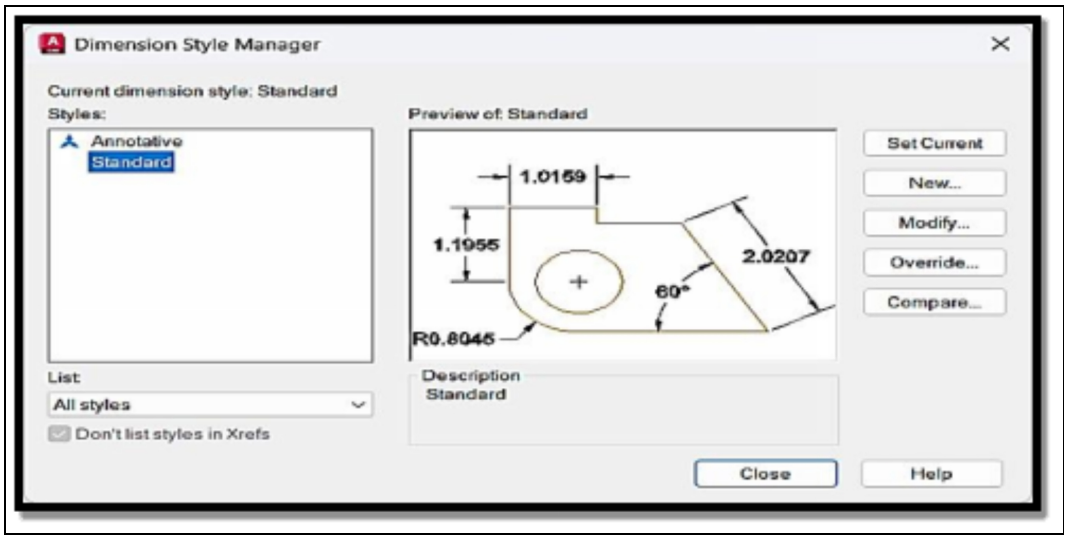
- Enable or disable the grid according to your preference. ***(NB: To ensure clarity, the figures here are displayed without the grid).***
- Navigate to the **Annotate tab** located on the ribbon. Here is the set of Annotate ribbon panels.

- To access the diagonal arrow on the Dimensions panel of the ribbon.




This arrow can be found on the right side of the Dimensions panel title bar. It is important to observe that the Text, Dimensions, Leaders, and Tables panels are equipped with arrows, commonly referred to as panel dialog box launchers.

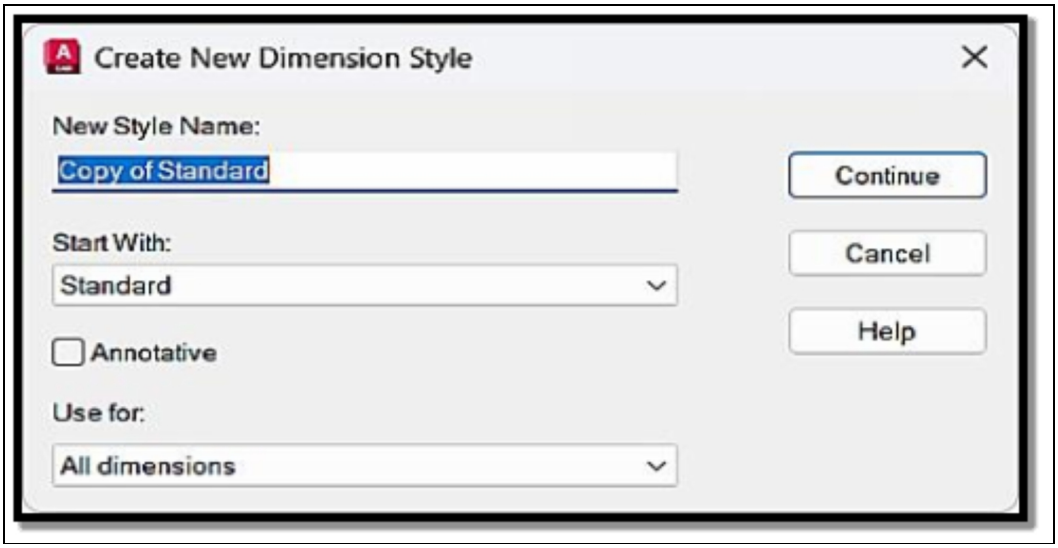
Opening the Dimension Style Manager will display the interface. The dimension style currently in use is called Standard. Similar to text styles, another pre-defined style in AutoCAD is the Annotative style. Annotative objects, such as dimensions, can be scaled to accurately display their size in paper layouts using the Annotative property. The distinction between the **Standard** and **Annotative** styles lies solely in the value assigned to the Annotative property.




Adjacent to the Styles list, you will find a preview that showcases a sample image illustrating numerous dimension variable settings of the style currently highlighted in the Styles list. The previewed style is also displayed at the top of the dialog box. The preview image is automatically updated whenever a change is made to a dimension setting. There is a Description

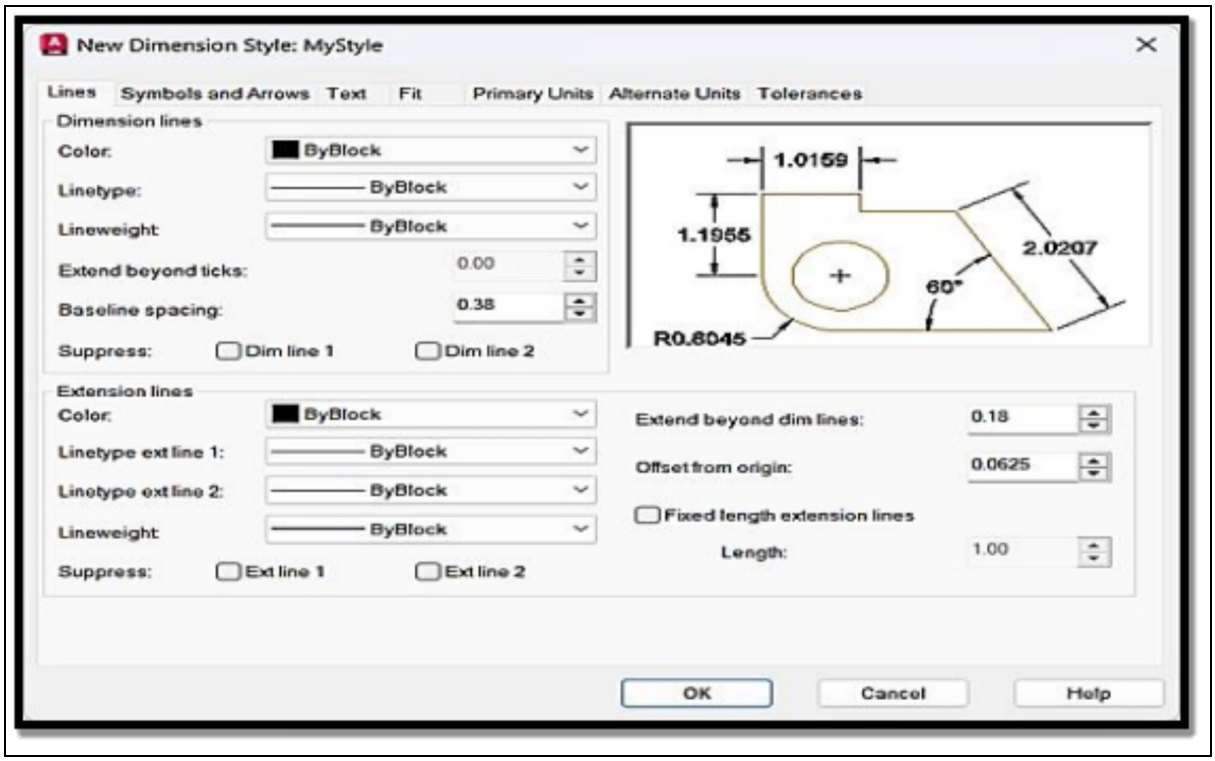
box located below the preview image. Currently, the description only states that the style is **Standard**.

- Ensure that the Standard option is selected in the Styles list. Next, you can create a new style by building upon the Standard style. Located to the right of the preview, you will find a series of five buttons. The second button in this set is known as the **New button**.
- To begin, locate and click on the "**New**" button. By selecting this option, you will be able to access the Create New Dimension Style dialog box.

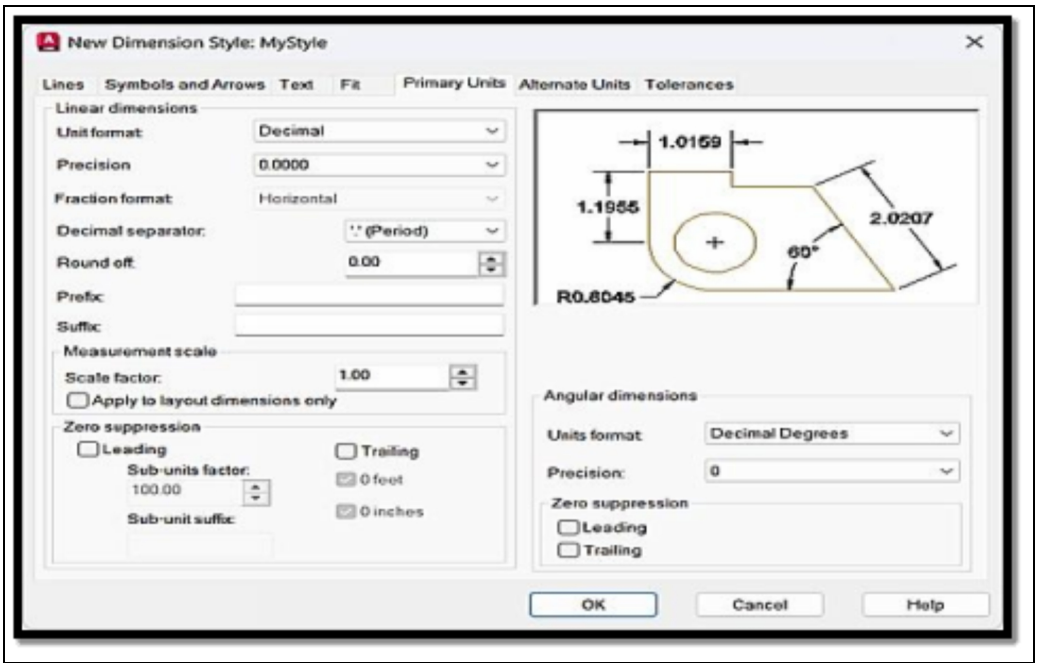



**If needed, simply double-click on the New Style Name edit box to select the text Copy of Standard.**

- Enter a name, like **MyStyle**, for the new dimension style, and then click **Continue**. Creating a new dimension style will lead you to the **New Dimension Style dialog box**. In AutoCAD 2025, you'll find seven tabs that give you full control over the dimension settings. These tabs cover everything from dimension lines and symbols to text, fit, primary units, alternate units, and tolerances.




- Navigate to the Primary Units tab. There are options in AutoCAD to adjust the units for both linear and angular measurements. The drop-down lists for Unit format and Precision in the Linear dimensions and Angular dimensions panels resemble the lists found in the Drawing Units dialog box. It's important to note that dimension units and drawing units are completely independent of each other. Each setting can be adjusted individually, and they can have different configurations. Ensure that the Unit format in the Linear dimensions panel is set to Decimal. Ensure that the Units format in the Angular dimensions panel is set to Decimal Degrees. Ensure that the values in your dialog box are up to date by making the necessary adjustments. To complete this exercise, you only need to modify the number of decimal places displayed in the Precision drop-down list on the Linear dimensions panel. The default value is set to **0.0000.** Set the value to **0.00**.




- Access the Precision drop-down list located in the Linear dimensions panel.
- Opening the precision settings will reveal a comprehensive list that spans from 0 to **0.00000000**.
- Choose **0.00** from the dropdown menu.

After closing the drop-down list, the selected precision will be displayed as 0.00. Observe the alteration in the preview image, which now displays decimals with two decimal places. Now, it's time to finish this step of the process by going back to the Dimension Style Manager. Before you exit the New Dimension Style dialog box, make sure to explore all the different tabs and take note of the extensive range of dimension variable settings that are available for adjustment. In addition to modifying dimension variables through the command line, you have the option to enter their names and input a new value. However, using the dialog box simplifies this process significantly, providing a more user-friendly experience. Furthermore, the preview image offers instant visual feedback when adjusting a dimension variable setting. In the Lines Symbols and Arrows tabs, you will discover a variety of options to modify the dimensions of your geometry, including adjusting its size and position. Within the Text tab, you can fine-tune the appearance and positioning of dimension text. In this tab, you can choose from a variety of text styles that have been pre-defined in the drawing. In the Fit tab, AutoCAD provides options for managing situations where there

is a tight fit that may cause ambiguity in the arrangement of dimension geometry and text. This is the location where you can modify the Annotative property. As an AutoCAD expert, you are already aware that Primary Units allow you to specify the units for displaying dimensions. With Alternate Units, you have the option to add a secondary unit specification alongside the primary dimension unit. As an AutoCAD expert, you can use dimension variable settings to conveniently showcase dimension values in both inches and centimeters. Dimension specifications are enhanced with tolerances to establish an acceptable range of values, such as in the case of a machining process. The Tolerances tab provides a variety of options for displaying tolerances.

- To save your changes and exit the New Dimension Style dialog box, simply click on the **OK button**.

In the Dimension Style Manager dialog box, you will notice that the newly created dimension style, MyStyle, has been successfully added to the list of available styles. By now, you should have defined the Annotative, MyStyle, and Standard dimension styles in your drawing. Ensure that you have selected the desired style and set it as the current style. You can easily identify this by looking at the line located at the top of the dialog box. It will display the name of the new style after the colon that follows the "**Current dimension style**". By selecting different styles from the Styles list, you will be able to view descriptions in the Description box that explain how these styles are connected to the current style. Give it a shot.

- Choose the **Annotative option** from the Styles list.

The description explains that Annotative is similar to the new style **MyStyle** but with a variable overall scale and different precision for dimensions and tolerances. The values for the overall scale, precision, and tolerance precision are set to **"0.00", "4",** and **"4"** respectively. On the other hand, when you designate Annotative as the active style, the description of the new style will be based on Annotative. Give it a shot.

- To set Annotative as the current style, simply click the **Set Current button** in the Styles list. The Description box now displays the annotative feature.

- Choose the **MyStyle option** from the Styles list. The description has been updated to indicate that the new style is similar to the Annotative style but with a fixed scale and precision of two decimal places. The values for Annotative, Overall Scale, Precision, and Tol Precision are now set to specific values.
- To set the new style, MyStyle, as the current dimension style again, simply click on **Set Current**.
- To exit the Dimension Style Manager dialog box, simply click on the **Close button**.
- When working on a template drawing, it is important to save your changes and close the drawing.

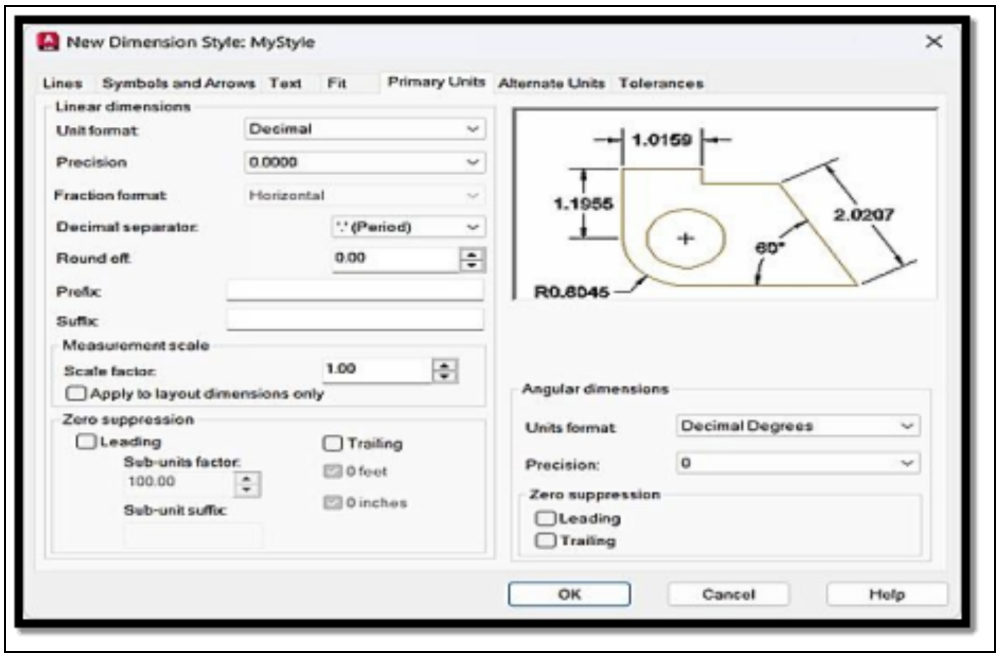
When creating a new drawing from the modified template, it is important to ensure that MyStyle is set as the current dimension style. The name of the dimension style can be found in the Dimension Style drop-down list located on the Dimensions panel of the ribbon.

# Creating Linear Dimensions in AutoCAD

AutoCAD offers a wide range of commands and features that greatly assist in the precise creation of dimensions. In this exercise, you will be creating some basic linear dimensions that will be formatted using the dimension style you have just created.

**Here's a helpful suggestion:**

- Select the **Linear tool** located in the Dimensions panel on the Annotate tab of the ribbon.
- Choose an object or indicate two origin points for the extension lines.
- Indicate the desired location for the dimension line.
- Before starting this exercise, create a triangle with dimensions (3.00, 4.00, 5.00), a vertical line of length 4.00, and a horizontal line of length 6.00 positioned above the center of the display.

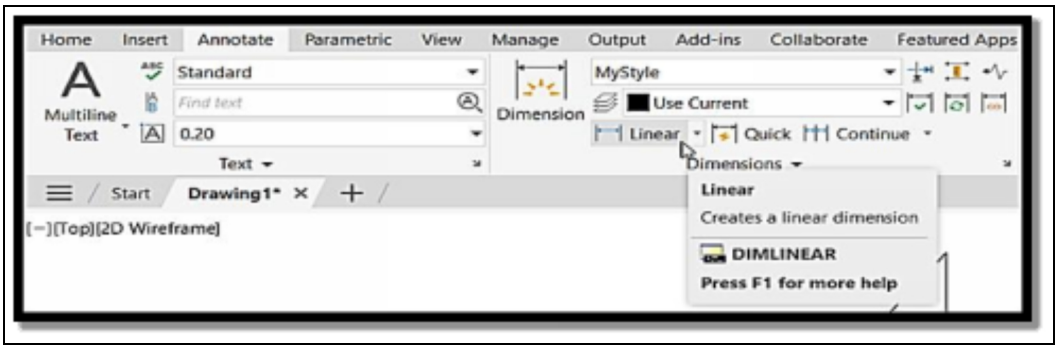



**Start by incorporating dimensions into the triangle.**

- To set the current layer for dimensions in your drawing, navigate to the Layers panel on the Home tab and select the desired layer from the Layer drop-down list.

The dimensioning commands in AutoCAD are highly optimized for maximum efficiency. Their full names can be quite lengthy. The dimension commands in AutoCAD typically start with "**DIM**" and are then followed by the specific name of the dimension type, such as **DIMLINEAR, DIMALIGNED, or DIMANGULAR**. Use the ribbon interface to save time and effort by avoiding the need to manually type out these names.
**Here's a helpful suggestion:** The Dimensions panel on the ribbon's Annotate tab includes a control for Dim Layer Override. This control allows you to specify the layer on which new dimensions will be created. You can choose a specific named layer or use the current drawing layer. If you consistently place your dimensions on a layer called "dims," you can easily make that layer the current one in this location.

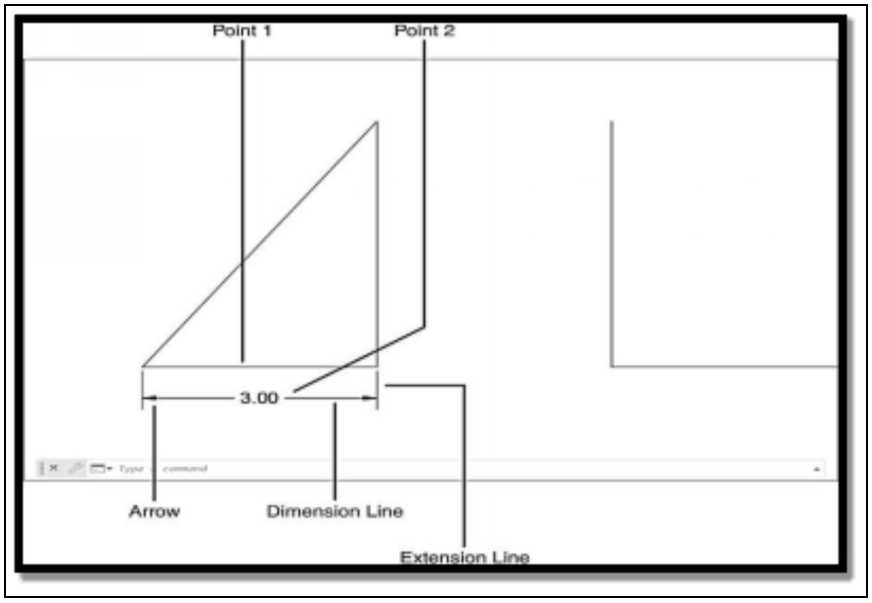
- Start by positioning a linear dimension beneath the triangle's base. To access the Linear tool from the Dimensions panel, navigate to the Annotate tab on the ribbon. Locate the tool in the panel.




To access the Linear tool, you may need to open the drop-down list on the ribbon. The last dimension command used will be placed at the top of the list for easy access. Initiate the **DIMLINEAR command** to begin. The command line will display the following prompt: Indicate the initial origin of the extension line or select an object. At this stage, there are two possible approaches to consider. There are two methods for determining the starting point of the extension lines and selecting the object to be dimensioned, allowing AutoCAD to automatically position the extension lines. In most basic applications, the second method is more efficient.

- Press the **Enter key** to enable object selection.

When using AutoCAD, the crosshairs are replaced by a pickbox and the software prompts you to make a selection. Choose the object you would like to dimension: Identify the horizontal line located at the base of the triangle.



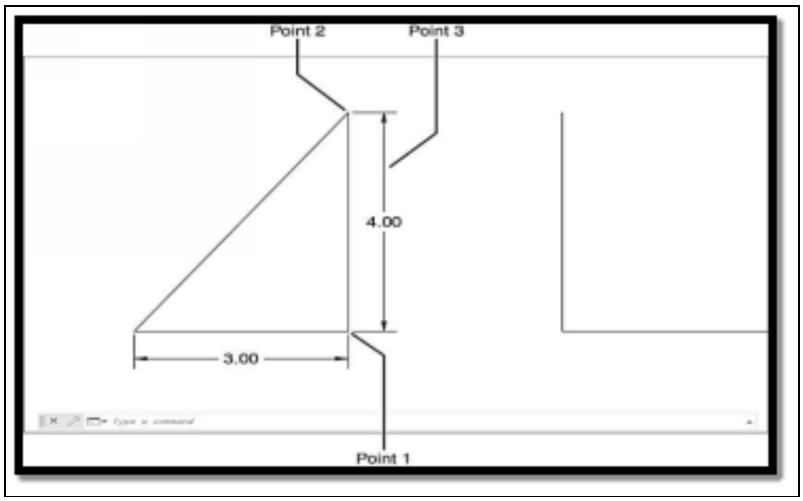
When using AutoCAD, a dimension can be created effortlessly. This includes extension lines, a dimension line, and text. You have the flexibility to easily move this dimension away from the selected line. In AutoCAD, the dimension line and text are positioned according to your instructions, while ensuring that the text remains centered between the extension lines.
**Here is the prompt:**
**Indicate the desired location for the dimension line or [Mtext/Text/Angle/Horizontal/Vertical/Rotated]:**
Locating the dimension line is a straightforward process in the default sequence. To modify the text content, you have a couple of options in AutoCAD. You can use the Mtext or Text command to make the changes directly, or you can use the DIMEDIT command to modify the text at a later time. You can use Angle, Horizontal, and Vertical to specify the orientation of the text. Linear dimensions in AutoCAD are typically set to display horizontal text by default. Rotated enables the user to effortlessly adjust the orientation of the entire dimension, causing the extension lines to extend at an angle away from the object being dimensioned. The text will remain in a horizontal orientation.

- Indicate a position approximately 0.50 units below the triangle.

Congratulations on completing your first dimension! Please note that the figures and illustrations in this chapter are displayed with a zoomed-in view of the relevant object to enhance clarity. Feel free to adjust the zoom level according to your preference. Now, carefully examine the dimension you have just drawn to understand its composition. It is worth noting that AutoCAD has conveniently positioned the extension line origins a small distance from the base of the triangle. To view this detail more clearly, it may be necessary to zoom in. The distance is determined by a dimension variable called DIMEXO. To adjust this distance, you can modify the DIMEXO value in the Modify Dimension Style dialog box. Look for the Offset from origin option on the Lines tab. Afterward, you can position a vertical dimension on the right side of the triangle. DIMLINEAR is capable of handling dimensions in both horizontal and vertical orientations.

- Execute the DIMLINEAR command once more.

Extension line origins are prompted in the same manner as before: Indicate the initial origin of the extension line or select an object: For this task, you need to manually indicate the starting point of the extension lines. Indicate the initial origin at the right-angle corner located in the lower right of the triangle.

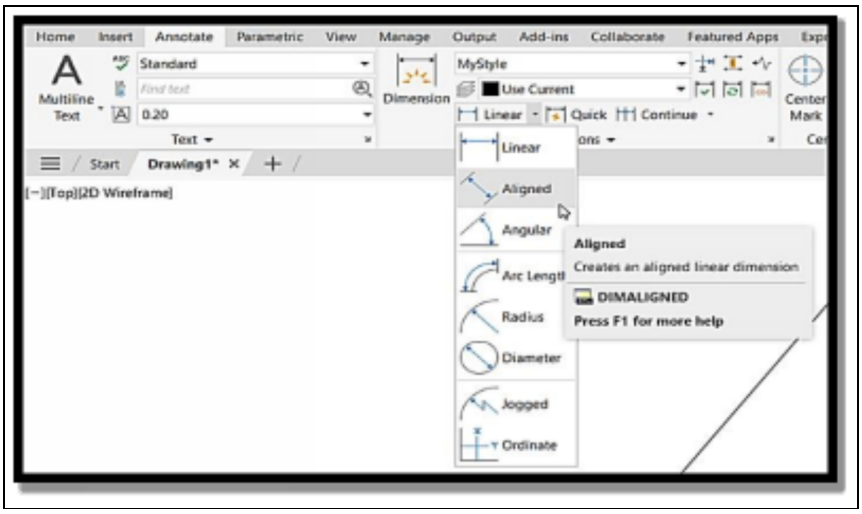



AutoCAD will require you to provide a second point:
Provide the origin for the second extension line:
Indicate a secondary origin at the upper intersection of the triangle.
The procedure remains unchanged from the previous steps. Ensure that you have a dimension ready to be effortlessly positioned, accompanied by the prompt:
Indicate the desired location for the dimension line or [Mtext/Text/Angle/Horizontal/Vertical/Rotated]:

- Indicate a position approximately 0.50 units to the right of the triangle. Ensure that the vertical dimension is incorporated into your drawing.

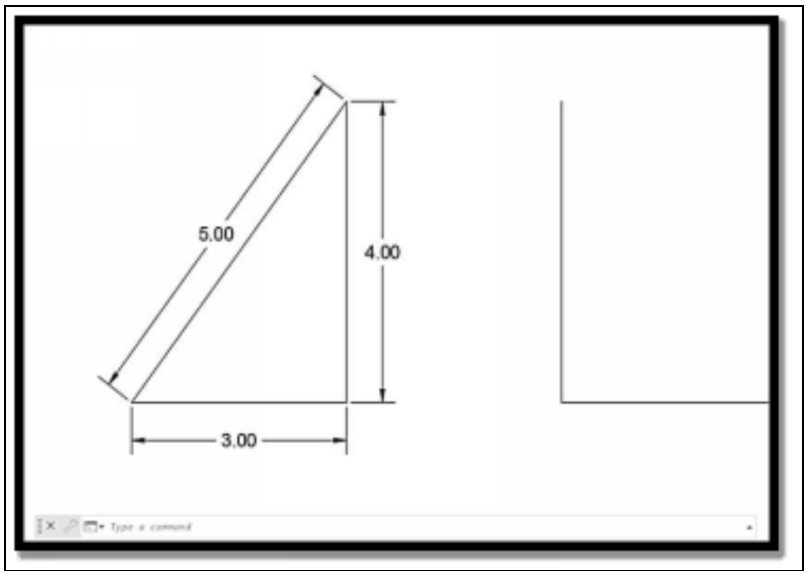
Next, we'll add a dimension to the diagonal side of the triangle. To accomplish this task, you will use the **DIMALIGNED command**. The Aligned dimension tool can be found in the drop-down menu that appears when you click the down arrow next to the Linear dimension tool.

- To access the Aligned tool, navigate to the **Annotate tab** and locate the **Dimension tool** drop-down menu on the Dimensions panel. Click on the tool.




- Press the **Enter key** to indicate your selection of an object. When using AutoCAD, you will be presented with the pick box and prompted to select an object for dimensioning.
- Identify the hypotenuse of the triangle.
- Indicate a position that is about 0.50 units above and to the left of the line.

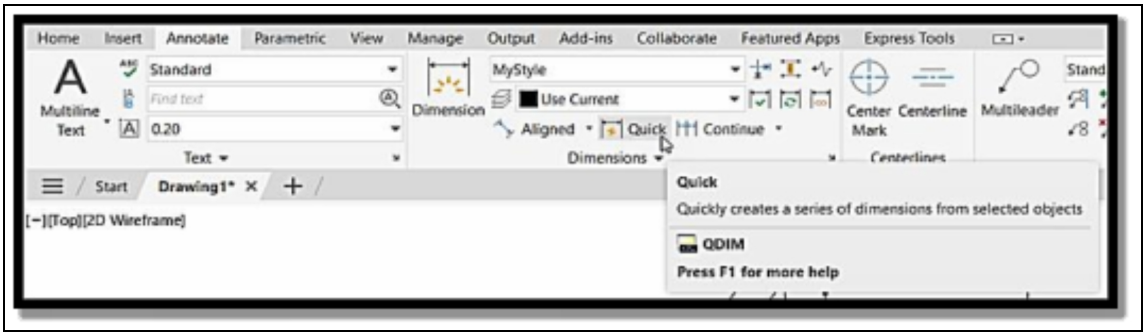
**It is important to note that AutoCAD automatically keeps horizontal text aligned and vertical dimensions as the default setting.**



# How to Draw Multiple Linear Dimensions Using QDIM

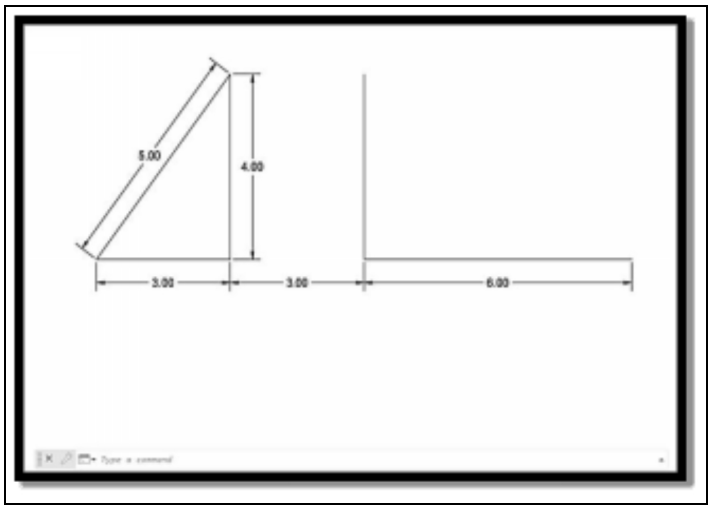
QDIM is a powerful tool that streamlines the process of generating multiple dimensions of specific types. Using this command, you can effortlessly generate a comprehensive set of interconnected dimensions in just a few simple clicks. When using QDIM, it is important to dimension the bottom of the triangle, the space between the triangle and the horizontal line, and the length of the line itself. This will ensure accurate measurements and proper documentation. Next, you will need to specify multiple points along the line to accurately dimension it. After making the necessary adjustments, you can modify the dimensions on the line to transition from a continuous series to a baseline series. In subsequent exercises, QDIM is used to generate various dimensions.

- Choose the lowermost point of the triangle and the horizontal line that extends 6.00 units to the right of the triangle. QDIM offers the convenient feature of noun/verb editing, enabling you to effortlessly select objects before initiating the command. The lines you have selected are visually emphasized and display interactive grips.
- To access the Quick tool, navigate to the Annotate tab on the ribbon's Dimensions panel.




**AutoCAD provides prompts:**
**Indicate the desired position of the dimension line, or [Continuous/Staggered/Baseline/Ordinate/Radius/Diameter/**
Adjusting the settings for continuous editing of datum points:

There are several different options available for multiple dimensions in this context. Continuous, Staggered, Baseline and Ordinate are all linear styles commonly used in AutoCAD. When it comes to dimensioning circles and arcs, Radius and Diameter are essential tools. The datumPoint command is used to modify the reference point for a series of linear dimensions. There are several functions in Edit that you can explore shortly. The settings in AutoCAD provide options for controlling how associated dimensions are created with the QDIM function. The continuous dimensions in AutoCAD are positioned end to end. When using AutoCAD, you can effortlessly generate and align three linear dimensions consecutively.




# Important

Dimensions or objects to be dimensioned can be selected for the QDIM command. When you choose objects, QDIM will generate fresh dimensions for them. On the other hand, if you choose dimensions, QDIM will modify or recreate those dimensions based on the options you choose.

- Indicate a position approximately 0.5 units below the triangle.
- Next, you can make adjustments to the horizontal dimension on the right to measure the line length at various lengths.
- Execute the QDIM command again.
- Choose the dimension located to the right and below the line labeled 6.00. It is important to note that this dimension can be selected individually. The three continuous dimensions were

created as individual objects, despite being created at the same time.

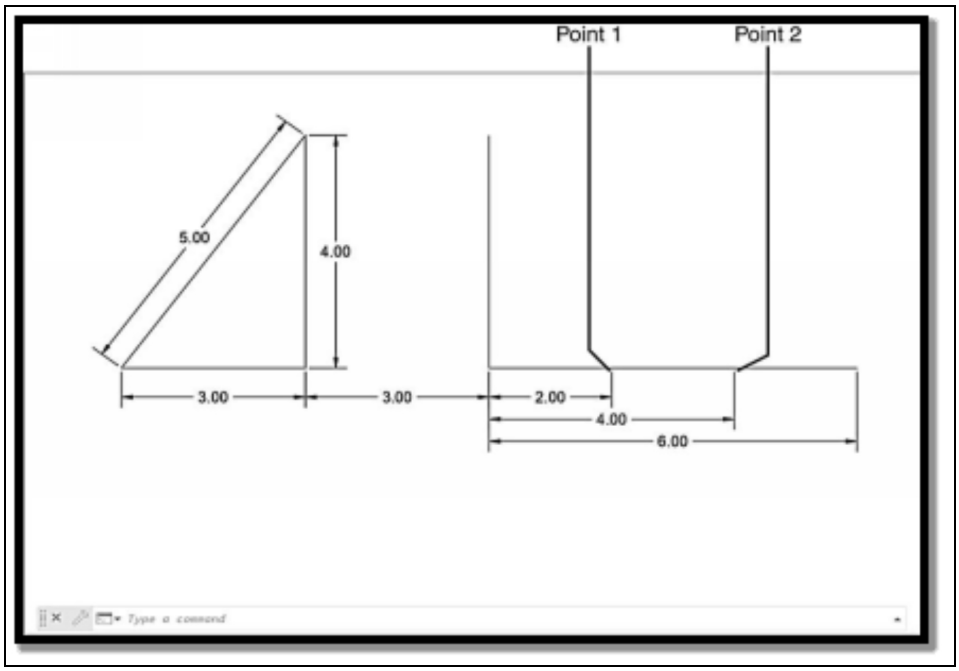
- To conclude object selection, simply perform a right-click.
- With AutoCAD, you can effortlessly position a dimension exactly where you want it. When a point is specified, the chosen dimension of 6.00 is recreated at that location. There is a way to make your work more engaging than simply recreating the dimension.
- Choose the **Edit option** from the command line.

When using the QDIM command, Xs will be automatically added at the two endpoints of the dimensioned line. You will then be prompted for further actions. Specify the dimension point you would like to remove or choose [Add/Exit] to exit: The Xs represent the dimension points. Additional points can be included to accurately dimension the line.

- Choose the "**Add**" option from the command line.

The prompt is modified slightly, as follows:
Specify the dimension point you would like to add or choose to remove or exit:

- Indicate the location of Point 1




The point is located 2 units away from the left endpoint of the line. The QDIM feature in AutoCAD may continue to prompt for points even after selecting the desired objects. This can be resolved by checking the settings and ensuring that the correct options are selected.

- Indicate the location of Point 2. The point is located 2 units away from the right endpoint of the line.
- Use the <**Enter**> key or the spacebar to complete the selection of endpoints.

The QDIM command now splits the original 6.00 dimension into a sequence of three consecutive 2.00 dimensions. Ensure that you have not overlooked any steps. Instead of using continuous dimensions, opt for the baseline type when drawing these three dimensions.

- Choose **Baseline** from the command line.

When using QDIM, the three dragged dimensions are instantly converted to a baseline form.

# Important

When using QDIM in AutoCAD, it's important to note that the software will remember the last option you used for drawing dimensions. This means that if you were to enter the command now, it would default to Baseline dimensions instead of Continuous dimensions. Indicate a point where the top dimension, which is the shortest among the three, is positioned approximately 0.5 units below the line.

# Using DIMBASELINE and DIMCONTINUE

**You can create baseline and continuous dimensions individually using separate commands. Here is a general procedure that is commonly used with these commands:**

- Create a preliminary linear dimension.
- Select the **Baseline or Continue tool** located in the Dimensions panel on the Annotate tab of the ribbon.
- Indicate the origin of the second extension line.
- Provide the origin for another extension line.
- Press the **Enter key** to exit the command.

# Important

Baseline and continued dimensions can be drawn using either the current dimension style or the style of the selected dimension if it differs from the current style. The behavior you're referring to is determined by the **DIMCONTINUEMODE** system variable. By setting the value to 0, which is the default, the current style will be used. However, if you set it to 1, the style of the selected dimension will be used instead.

# Drawing Ordinate Dimensions

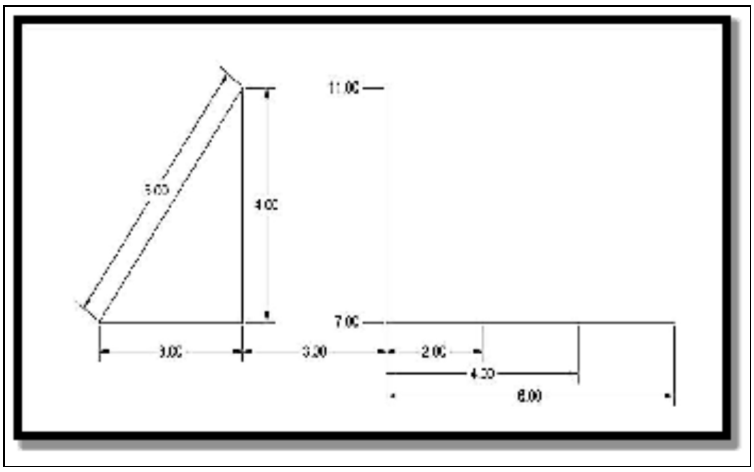
Ordinate dimensions provide an alternative method for specifying linear dimensions. These measurements are commonly displayed to indicate various horizontal and vertical distances originating from a specific point, typically the corner of an object within a design. Efficiency is achieved by representing these dimensions as x and y displacements from a single point of origin, as they easily fit into a coordinate system. An ordinate dimension

is a type of dimension that is determined based on a fixed point of origin, rather than by directly measuring the objects being dimensioned. There are two methods for generating ordinate dimensions in AutoCAD. In AutoCAD, points are typically defined concerning the origin point (0,0) in your drawing. With QDIM, it is possible to designate a fresh datum point that acts as the starting point for a group of ordinate dimensions. When using **DIMORDINATE** in AutoCAD, you will need to temporarily relocate the coordinate system's origin to the desired point for specifying dimensions. In this exercise, you will become proficient in using both methods.

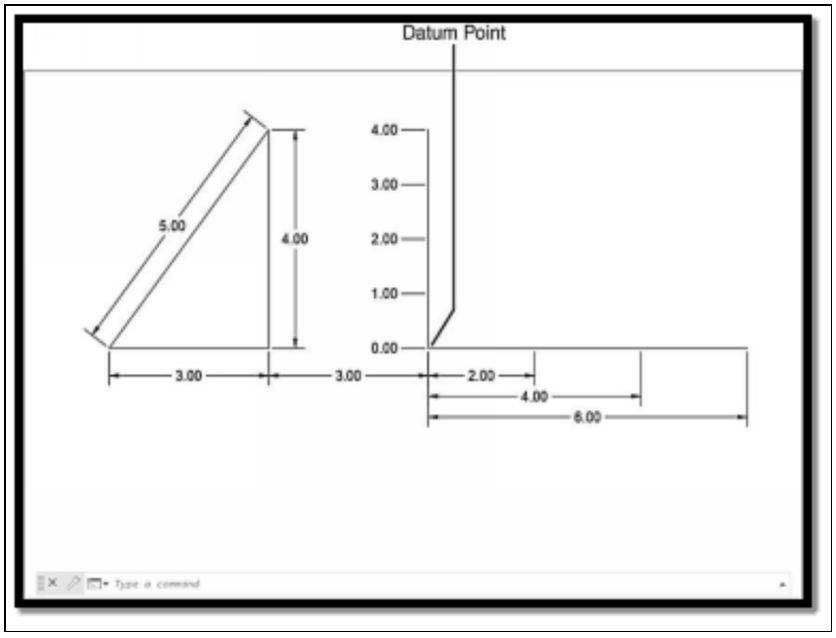
# Understanding QDIM and the Datum Point Option

Ordinate dimensions are used to specify a sequence of horizontal and vertical distances originating from the intersection of the two lines, extending to the right of the triangle. To add ordinate dimensions along the 4.00 vertical line, you can use the QDIM command. To add ordinate dimensions to the 6.00 horizontal line, you can use the **DIMORDINATE** command.

- To access the Quick tool, navigate to the Dimensions panel on the Annotate tab of the ribbon and click on it.
- Choose the vertical line.
- Use the right-click function to complete the selection of objects.
- Access the **Ordinate command** from the command line.
- Indicate a position approximately 0.5 units to the left of the line.




The dimensions displayed at the endpoints of the line indicate the y value of each endpoint. When using the ordinate option of QDIM or DIMORDINATE in AutoCAD, the software intelligently selects the appropriate x or y value based on the object you have selected. Since the values are measured from the origin at the lower-left corner of the grid, they may not be very practical. Another frequently employed application involves measuring points at the intersection of the two lines. To complete this set of dimensions, you can utilize the QDIM tool. Simply go back into QDIM, choose a new datum point, and make adjustments by adding or removing dimension points.




- Execute the QDIM command again.

- Choose the ordinate dimension at the bottom.
- Choose the highest ordinate dimension. It is important to note that these elements must be individually selected.
- Use the right-click option to finalize the selection of your geometry.
- Access the **Ordinate command** from the command line.
- Choose the **datum point** from the command line.

# QDIM prompts

**Choose a new datum point:**

- Identify the point where the two lines intersect. The establishment of the new datum point has been completed. Before exiting QDIM, you can add three additional dimension points and remove one if needed.

- Choose the **Edit option** from the command line.
- Choose the "**Add**" option from the command line.
- Indicate the points located 1.00, 2.00, and 3.00 units above the bottom of the vertical line. When specifying the points to add, they will be marked by Xs.
- Choose the "**Remove**" option from the command line.
- Eliminate the point where the two lines intersect.
- Press the **Enter key** or the spacebar to conclude the specification of points.
- Indicate a point approximately 0.5 units to the left of the vertical line, just like you did previously.

It is important to observe that the dimension at the top is automatically adjusted to accurately represent the updated datum point. Afterward, you can dimension the horizontal line by using the **DIMORDINATE command**.

# Understanding DIMORDINATE and the UCS Command

To create a series of ordinate dimensions above the horizontal 6.00 line, you can utilize the DIMORDINATE command. To create a new origin for the coordinate system, you can utilize the UCS command. Understanding user coordinate systems is crucial when working with 3D drawings, although we won't delve into them extensively in this context. While DIMORDINATE may only create one dimension at a time, it does offer certain advantages compared to the QDIM system. Firstly, there is no need to manually edit the dimension to add or remove points. In addition, you can generate a wide range of leader shapes.

- To make the UCS icon visible, you can easily turn it on by typing "**ucsicon**" followed by the **Enter key**, and then selecting the "**On**" option.
- Enter "**UCS**" and press the **Enter key**. The UCS command is executed, resulting in a visually striking dashed yellow rubber band that extends from the lower-left corner of the drawing area or grid origin to the current position of the crosshairs. The default method for specifying a new coordinate system in AutoCAD is by moving the point of origin. This is the simplest option available in the UCS command. AutoCAD provides prompts:

Indicate the origin of the User Coordinate System (UCS) or [Entity/Designation/Element/Previous/Perspective/Global/X/Y/Z/Axis]

- Indicate the point where the two lines intersect.

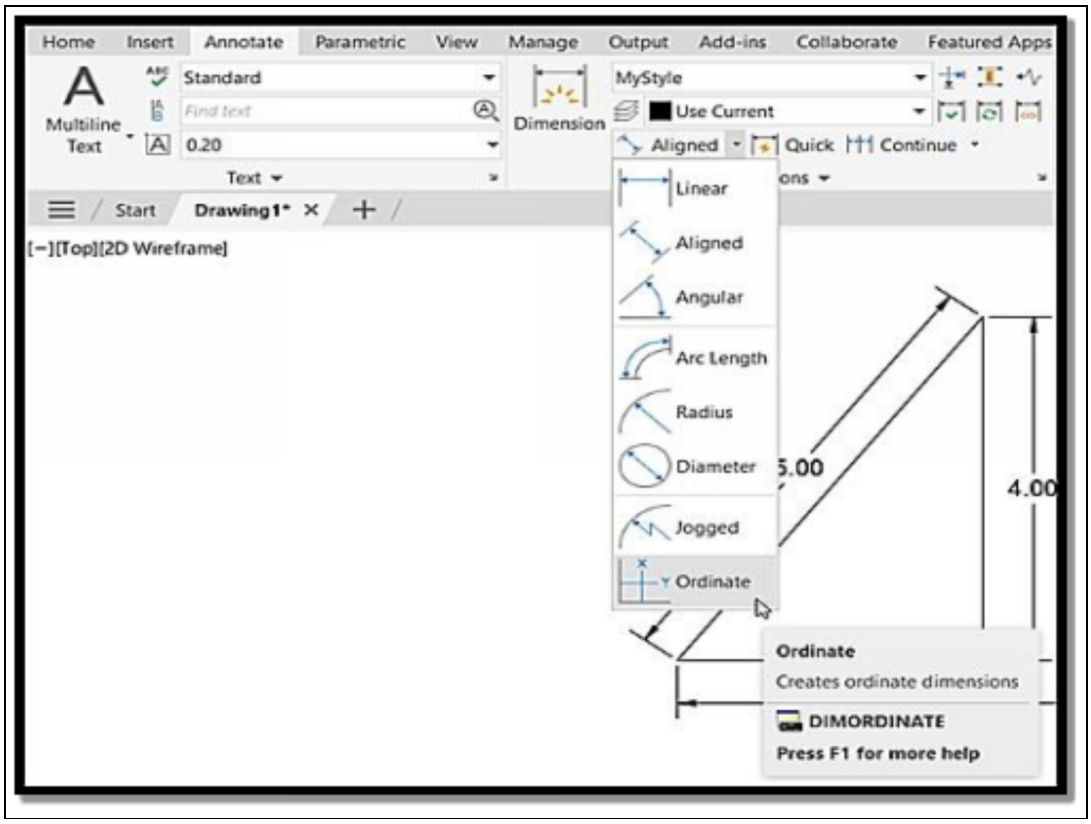
# AutoCAD prompts

**Indicate a point on the X-axis or use the colon symbol to specify:**

- Rotate the cursor in a deliberate circular motion around the intersection. The previewed axis is visible in the newly specified coordinate system. When a third point is specified, it will automatically be assigned as a point on the new axis. By

pressing <**Enter**> after completing the command, the orientation of the current coordinate system will be preserved.

- Press the **Enter key** to confirm the new origin. The grid's origin point is relocated, causing the lower-left corner to move accordingly. By positioning your cursor at the intersection and observing the coordinate display, you will notice that this point is currently indicated as (0.00,0.00,0.00). When the user coordinate system icon is enabled and configured to relocate to the origin, it will automatically shift to the specified location. In addition, when the grid is enabled, the origin point automatically adjusts to the new position.
- Go to the **Annotate tab** and click on **Dimension**.

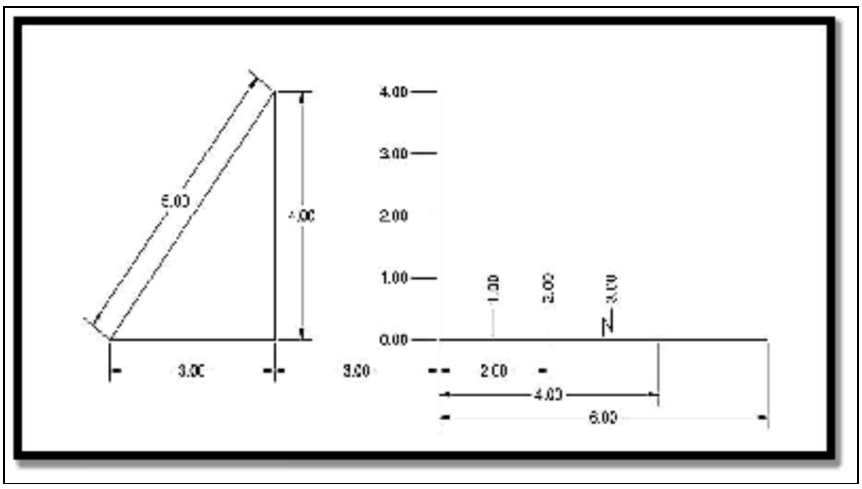



- Select the **Ordinate dimension tool** from the drop-down menu, as indicated.

AutoCAD provides prompts:
Indicate the precise location of the feature:

Simply specify a point and an endpoint for a leader. The dimension text will indicate either an x or a y displacement from the origin of the current coordinate system, depending on the location of the endpoint to the first point.

- Indicate a point along the 6.00 line, 1.00 to the right of the intersection.




Based on the given information, the point will be located at (1,0) in the new coordinate system.

# AutoCAD prompts

**Non-associative dimension created Specify leader endpoint or [Xdatum/Ydatum/Mtext/Text/ Angle]:**

- Ordinate dimensions created in this manner are nonassociative. Next, you'll need to indicate the endpoint for the leader. To specify the display of the dimension text for the x or y coordinate, simply enter *'x'* or **'y'**. When selecting the endpoint accurately, AutoCAD will automatically choose the correct coordinate for you. If you wish, you can input your text. However, doing so would undermine the objective of establishing a coordinate system that automatically calculates the distances from the intersection of the two lines.
- Indicate an endpoint that is positioned 0.50 units above the line. Make sure to incorporate the 1.00 ordinate dimension.

- To achieve the desired result, you can use the **DIMORDINATE command** once again. This time, you will need to specify the second ordinate dimension at a distance of 2.00 units from the origin point. The coordinates for this point are (2,0).
- Let's go over DIMORDINATE one more time.
- Indicate a point along the line that is 3.00 units away from the origin. The coordinates of the point are (3,0).
- Use your cursor to navigate left and right and observe the various leader shapes generated by **DIMORDINATE**, which are determined by the endpoint of the leader.
- To create a broken leader, you can specify an endpoint slightly to the right of the dimensioned point. After completing your task, it is important to switch back to the world coordinate system. This is the standard coordinate system that you have been using throughout.
- Enter the command "**UCS**" and press **Enter**.
- Choose "**World**" from the command line.

By performing this action, the origin is reset to its initial location at the lower left corner of your drawing window.

# How to Draw Angular Dimensions

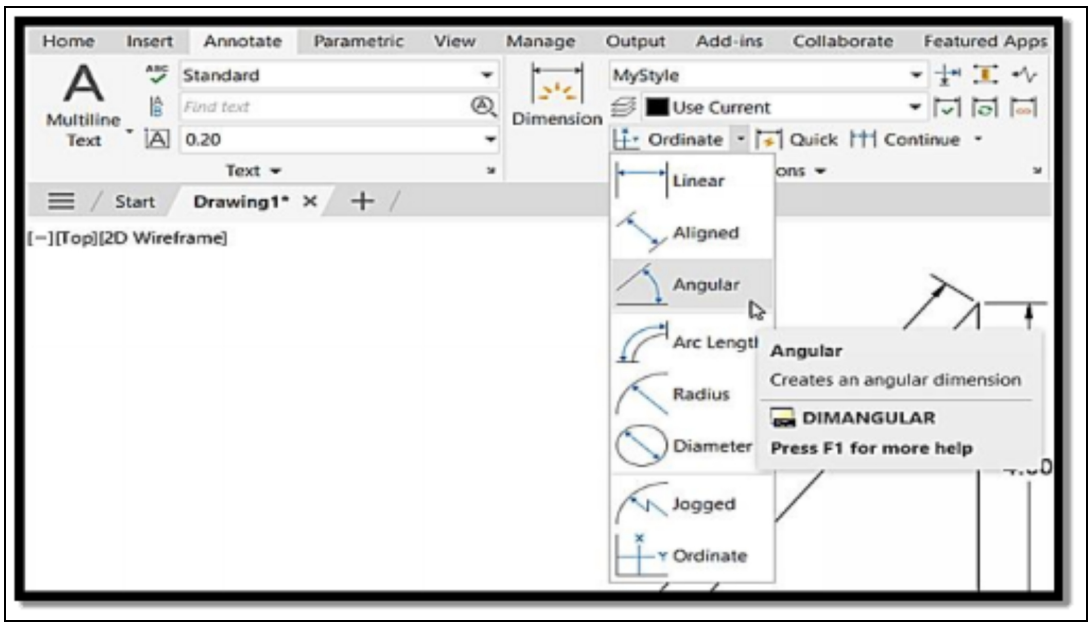
Angular dimensioning functions similarly to linear dimensioning, with the added step of selecting objects that create an angle. When using AutoCAD, the software will calculate the angle for you based on the geometry you choose. This can include two lines, an arc, a portion of a circle, or a vertex and two points. AutoCAD will then create extension lines, a dimension arc, and text to indicate the angle.

**Here's a helpful suggestion:**

- To access the Angular Dimension tool, navigate to the Annotate tab and select **Dimensions**. From there, choose the **Dimension option** and then select **Angular**.
- Choose two lines that intersect to create an angle.
- Indicate the desired position for the dimension.

For this exercise, let's go back to the triangle and enhance it by adding angular dimensions to two of the angles.

- To access the Angular tool, navigate to the Annotate ribbon tab. From there, locate the Dimensions panel and open the **Dimension tools** drop-down menu. Finally, select the **Angular tool**.




**Here is the initial prompt:**
*Choose between arc, circle, line, or specify the vertex:*
With the DIMANGULAR command in AutoCAD, you can easily dimension angles that are formed by arcs, portions of circles, and lines. It's a handy tool for accurately measuring and documenting angular dimensions in your drawings. To manually specify an angle, simply press <**Enter**> and select the vertex and a point on each side of the angle. Start by selecting the lines.

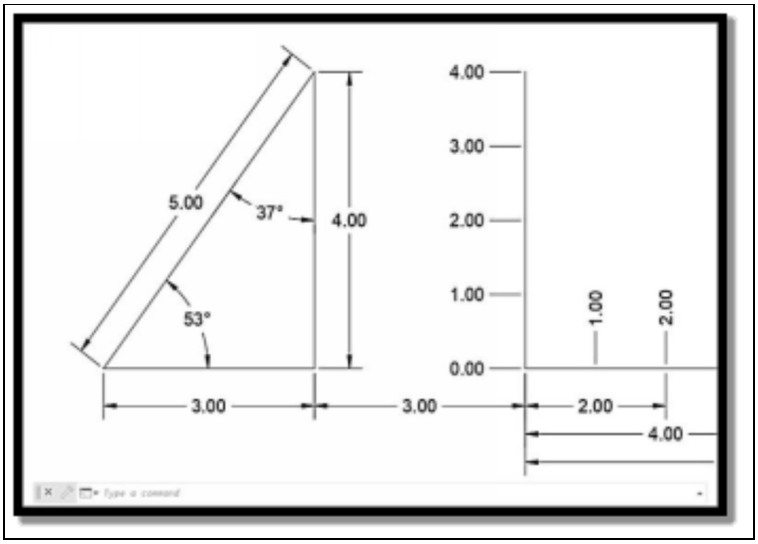
- Indicate the base of the triangle.

Provide the input for the next line:
Choose the second line:

- Choose the hypotenuse.

In AutoCAD, you can easily manipulate the dimension line by dragging it to the desired position, just like you would with linear dimensioning. For the dimension arc location, you have the flexibility to modify both the text and the text angle as per your requirements.
*Specify dimension arc line location or [Mtext/Text/Angle/ Quadrant]:*

- Move the cursor to observe the various possibilities for placing the dimension. Next, indicate a specific point between the two lines that have been selected.




**Ensure that the lower-left angle of your triangle is dimensioned. It is important to note that the degree symbol is automatically included in the angular dimension text.**

- Now, to dimension the upper angle, you'll need to specify its vertex.
- Execute the **DIMANGULAR command** again.
- Hit the **Enter key**. When using AutoCAD, the software will request the input of an angle vertex.
- To enable Snap Mode, simply click on the **Snap Mode button** if needed.
- Designate the point where the triangle's top intersects as the vertex of the angle.

AutoCAD provides prompts:
Indicate the endpoint of the first angle:

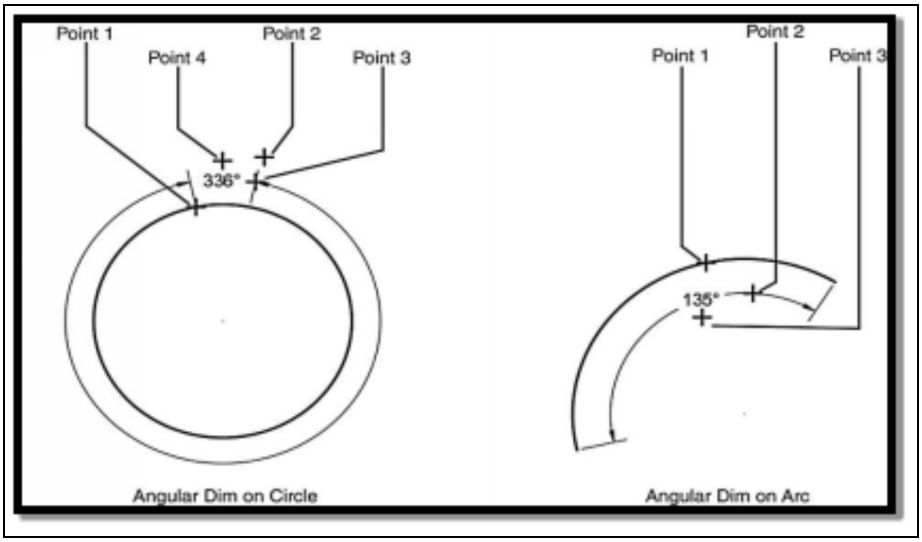
- Indicate a point along the hypotenuse.

This should be a snap point for accuracy. The most reliable option would be the lower-left corner of the triangle. AutoCAD provides prompts:
Provide the coordinates for the second endpoint of the angle:

- Indicate a point on the vertical side of the triangle. Ensure that all points along the vertical line are set as snap points.
- Ensure that the cursor is moved in a controlled manner, both upwards and downwards, while staying within the boundaries of the triangle. Observe how the arrows in AutoCAD are positioned outside the angle when you reach the vertex, resulting in a cluttered appearance. It is important to observe that when you move beyond the angle, AutoCAD will automatically switch to the outer angle.
- Indicate the desired position for the dimension arc

# Angular Dimensions on Arcs and Circles

In addition, it is possible to add angular dimensions to arcs and circles. AutoCAD generates extension lines and a dimension arc in both scenarios. When dimensioning an arc using an angular dimension in AutoCAD, the center of the arc is automatically assigned as the vertex of the dimension angle. The endpoints of the arc are then designated as the starting points for the extension lines. When working with circles in AutoCAD, the dimension line origins are determined by the point used to select the circle and a second point, as prompted by the software.



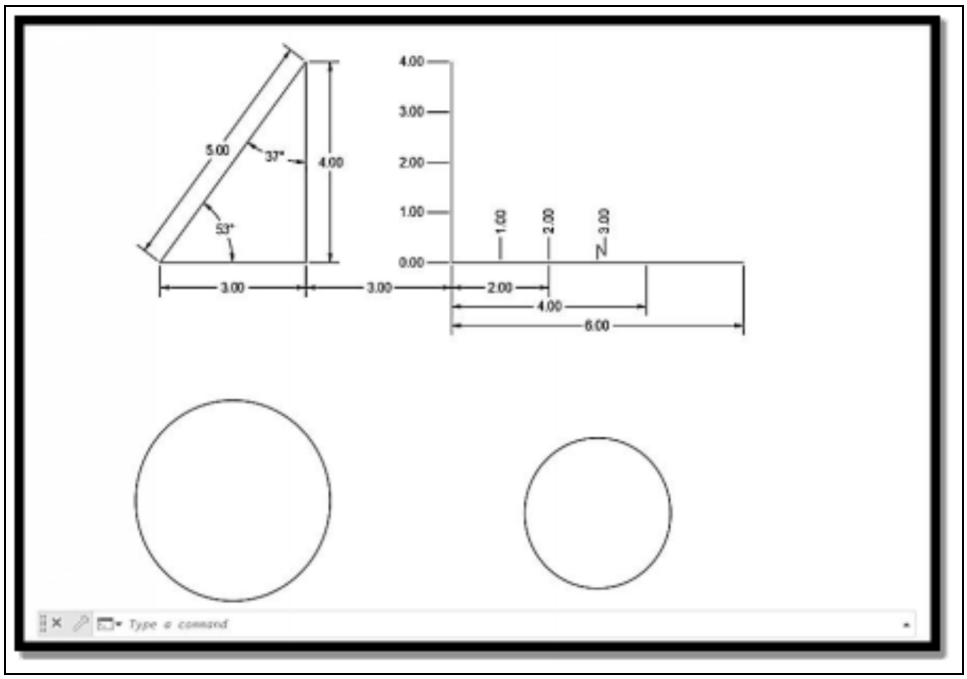

# About Dimensioning Arcs and Circles

Dimensioning circles and arcs follows a straightforward process, similar to the ones we've already discussed. It can be challenging at times when AutoCAD does not position the dimension where desired. Controlling the placement of text can be easily achieved by adjusting the dimension variables. For this exercise, you will be creating a center mark along with diameter and radius dimensions.

**Here's a helpful tip:**

- To access the radius or diameter dimension options, navigate to the Annotate tab and select Dimensions. From there, choose the **Dimension tool** and select either **Radius or Diameter**.
- Choose either an arc or a circle.
- Provide the desired location for the endpoint of the leader line.

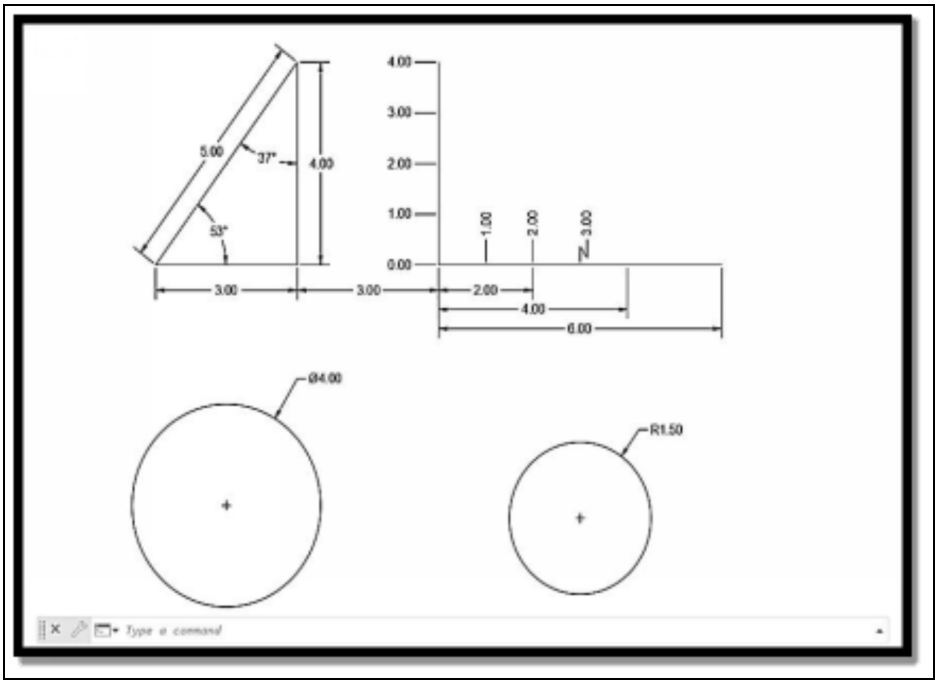
Before commencing this exercise, it is necessary to create two circles, each with a radius of 2.00 and 1.50. When dealing with drawings that have multiple layers, it is important to avoid drawing circles on the dimension layer.



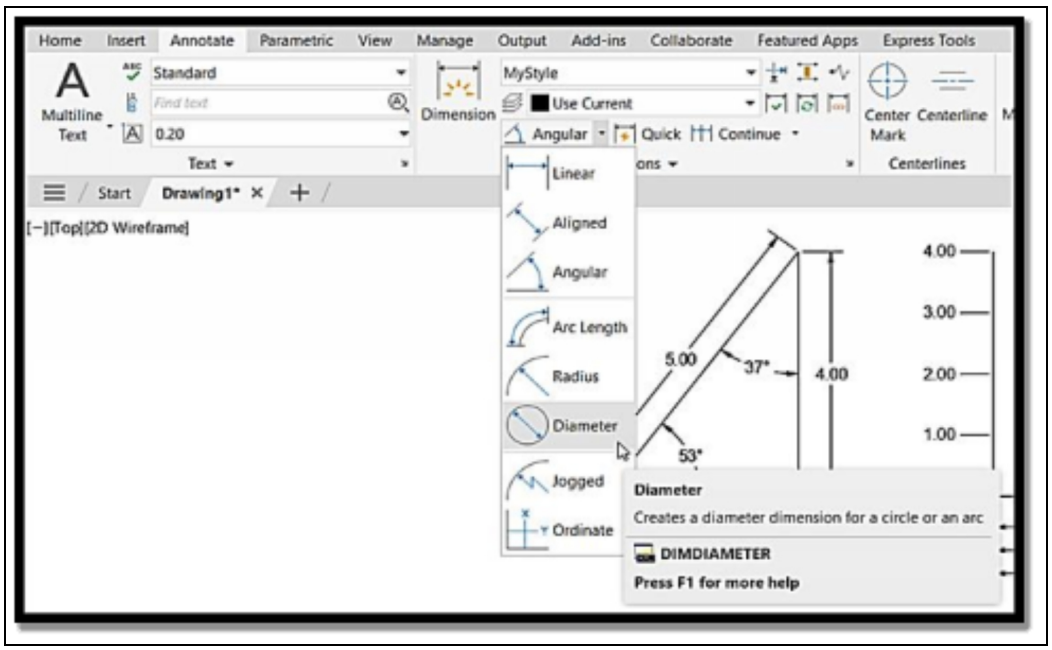

- If needed, switch back to the dimension layer.

# Diameter Dimensions

To add a diameter dimension to the circle with a radius of 2.00 on the left.



- To access the Diameter dimension tool, navigate to the Annotate menu, then select **Dimensions,** and finally **Dimension**.




AutoCAD provides prompts:
Choose between an arc or a circle:

- Choose the larger circle.

When using AutoCAD, you may come across a situation where a diameter dimension is displayed and the software prompts you to specify the location of the dimension line. Indicate the desired location for the dimension line: [Mtext/Text/Angle]. You can easily modify the dimension text or adjust its orientation using the Text and Angle options. By manipulating the cursor, you can precisely position the dimension line within or around the circle.

- Choose a suitable position for the dimension line near the upper portion of the 2.00 circle.

It is important to note that the diameter symbol prefix and the center mark are automatically added by default.

# Radius Dimensions

The process of placing radius dimensions is identical to that of diameter dimensions, and the visual outcome is indistinguishable. There are slight variations in the text's radius value and the use of "**R**" instead of the diameter symbol.

# Create a radius dimension on the smaller circle

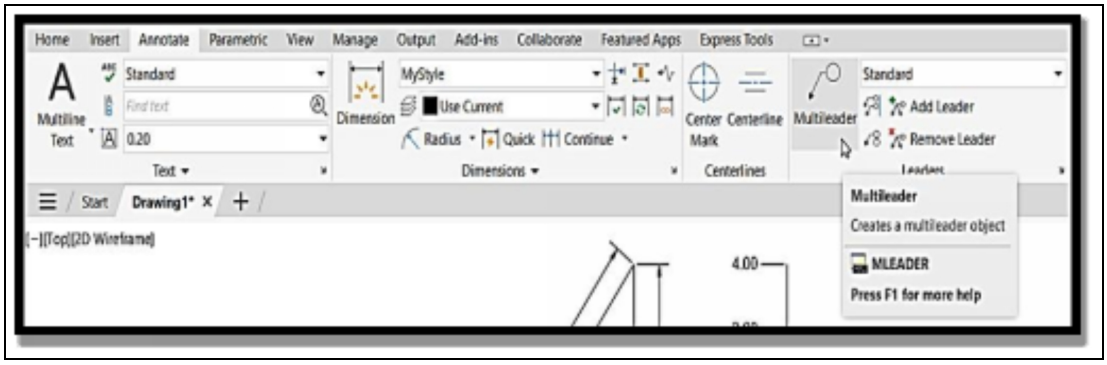
- Select the **Radius tool** from the Dimension tools drop-down menu located on the Annotate tab of the ribbon.
- Choose the smaller circle with a diameter of 1.50.
- Move the cursor within and outside the circumference of the circle.
- Choose a suitable position for the dimension line to finalize the dimension. Radius and center marks are automatically added in AutoCAD.

# How to Annotate with multileader

Radius and diameter dimensions, as well as ordinate dimensions, utilize leaders to link the dimension text to the object being dimensioned. You can create leaders or multileaders independently to attach annotations to various objects. Unlike other types of dimensions, a leader is a visual connection between an object and its annotation. It consists of a line or series of lines with an arrow at the end. When creating a leader in AutoCAD, the software does not automatically recognize or measure any selected object or distance. Before starting, it is essential to have a clear understanding of the dimension text or annotation you intend to use.

A multileader is a type of object that includes an arrowhead and one or more leaders that connect annotation text or symbols to annotated objects.
**Here's a helpful suggestion:**

- Click on the **Multileader tool** located in the Leaders panel on the Annotate tab of the ribbon.
- Indicate the location of the leader arrowhead.
- Indicate the desired location for the leader landing.
- Enter the dimension text.
- To annotate the circle on the right, start by adding a multileader with attached text. When it comes to dimension objects, multileaders may look similar to leaders, but they have their own set of commands, styles, and procedures that are unique to them.
- To create a multileader, go to the **Annotate tab** and select **Leaders** from the dropdown menu. Then, choose **Multileader**.




AutoCAD will request the location of the arrowhead:
Indicate the desired location for the leader arrowhead or enter the text/leader beforehand.

Choose from the following options:
Leaders are composed of an arrowhead, a leader line, and a brief landing line that links the leader to the text or annotation symbol. You can modify the sequence in which these elements are defined through the available options. Use the **Nearest object snap** to conveniently attach the arrow to the circle.

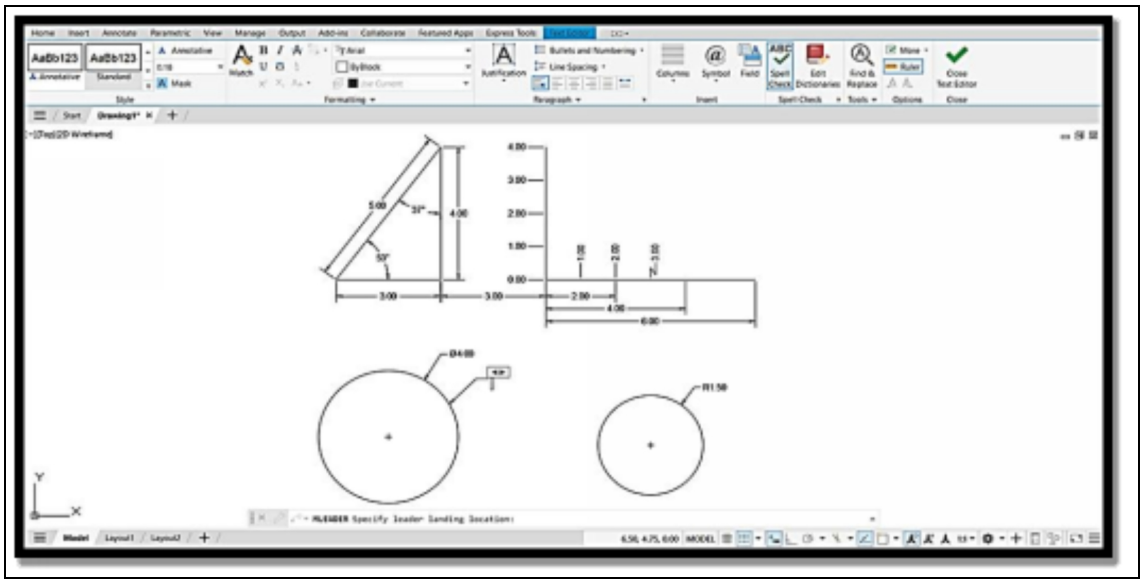
- To open the Object Snap shortcut menu, simply hold down the **Shift key** and right-click your mouse.
- Choose the "**Nearest**" option from the shortcut menu.

When you click, the Nearest Object snap feature enables you to effortlessly snap to the closest point on an object. If you haven't had the chance to explore this object snap feature, now would be a good time to familiarize yourself with it. While navigating through the drawing, the Nearest object snap marker will become visible as the crosshairs get closer to an object. When the crosshairs are positioned on an object, the Nearest tooltip label will replace the dynamic input prompt.

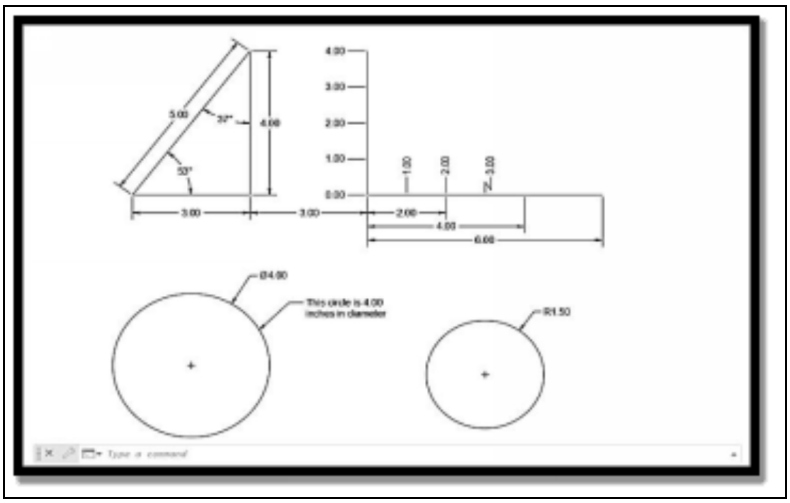
- Indicate the location of the leader arrowhead on the upper-right side of the larger circle with a radius of 2.00 units.
- Indicate the desired position for the leader landing, approximately 45° in angle and 1.00 units above and to the right of the initial point.

Remember to always double-check your measurements and dimensions before finalizing your design. This will help ensure accuracy and prevent any costly mistakes. Use polar tracking to accurately depict the angle of a leader line. By implementing this approach, your leaders will have a uniform and professional appearance. To enable polar tracking, you have two options. You can either click on the Polar Tracking button located on the status bar or simply press <**F10**>. The **Text Editor contextual tab** is displayed on the ribbon. There are two arrows located above the text input area that enable you to set the width of the text you are about to enter.

- Place your cursor inside the small gray box that has two arrows.



- Use the in-place text editor to effortlessly adjust the width of the box by clicking and dragging the arrows. Move it approximately 2.50 units to the right and release the mouse button.
- Input the diameter of the circle as 4.00 inches and press the **Enter key**.
- To finalize the command, simply click outside the in-place text editor.




Afterward, you can incorporate a multileader object along with an annotation symbol onto the smaller circle.

**Here's a helpful suggestion:**

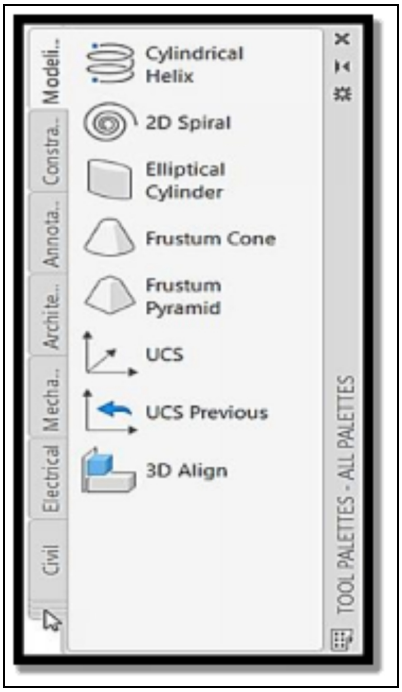
The MLEADER command allows for the conversion of an MText object into a multileader object, with the chosen MText object serving as its

content. To convert an existing MText object to a multileader, you can follow these simple steps:

- Begin the **MLEADER command**.
- Choose the "**Content first**" option, followed by selecting the "**Mtext**" suboption.
- When you are prompted, simply choose the MText object that you want to convert.
- Indicate the precise position of the arrowhead.

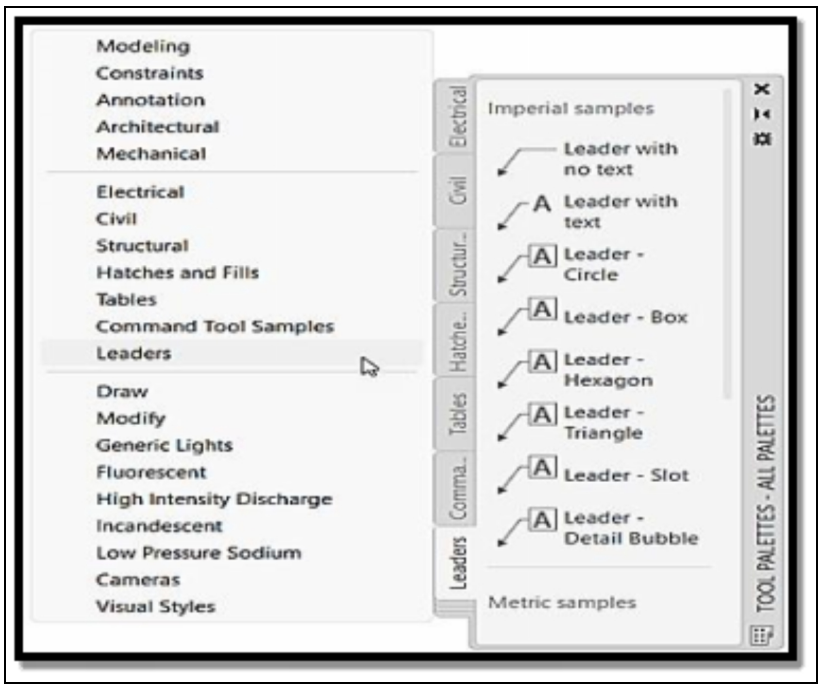
# The Leaders Tool Palette

In this section, you will learn how to add two leaders with tag annotations and utilize advanced tools for aligning and grouping multileader objects. At this point, you will utilize a tool palette for the first time. A tool palette is a tabbed collection of tools. To access the Tool Palettes window, there are multiple options available on the ribbon or the menu bar. However, in this case, you can conveniently utilize a keyboard shortcut. The tool palette is a convenient collection of tools organized in a tabbed format. Use the keyboard shortcut <**Ctrl**>+3. By selecting this option, you will have access to a comprehensive array of tool palettes.

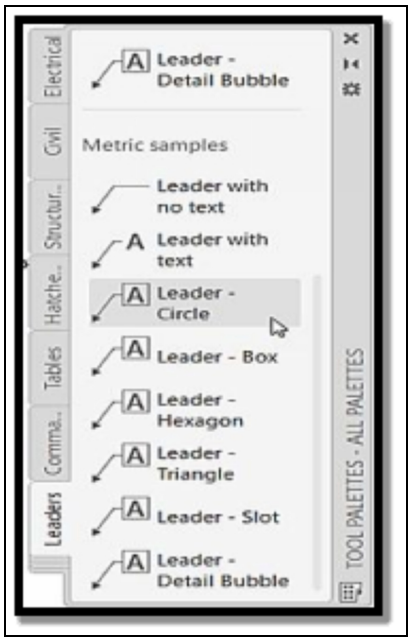


Take note of the numerous tabs positioned along the left (or right) side of the palette. Located at the lower section of these tabs is a compact area resembling stacked tabs. To access additional tool palettes, simply click on this area and a list will appear.

- Click on the designated area located at the bottom of the tool palette tabs, precisely below the tabs.




- Choose **leaders** from the provided list.




This palette includes two sets of leader styles: imperial styles located at the top and metric styles positioned at the bottom. There are a total of eight items in each list, and the lists are the same except for some differences in size. In this case, the metric Leader-Circle style tool is utilized.

- Select the Leader–Circle style tool from the Metric samples section of the tool palette.

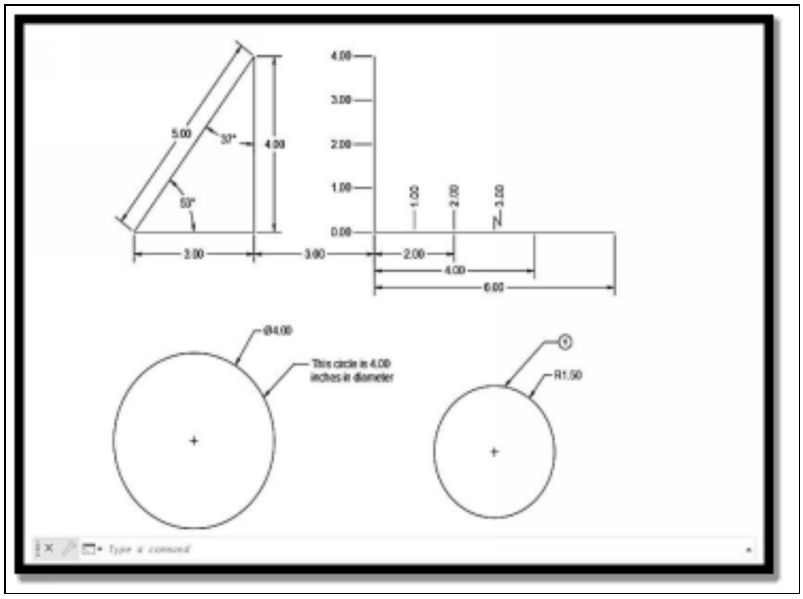
AutoCAD will prompt you to specify the location or option for the arrowhead, just like it did previously. Indicate the desired location for the leader arrowhead. [Enter text/leader first/Landing content first/Options]
**Here are the options:**

- With the **Leader–Circle style tool,** you can assign a number to the circle with a radius of 1.50. This leader is not dimensioning anything. It's just a tag that can be used to highlight or label an object in a drawing.
- To open the Object Snap shortcut menu, simply hold down the **Shift key** and right-click your mouse.
- Choose the "**Nearest**" option from the shortcut menu.
- Position the cursor over the top-right corner of the circle with a radius of 1.50 units and proceed to select it.

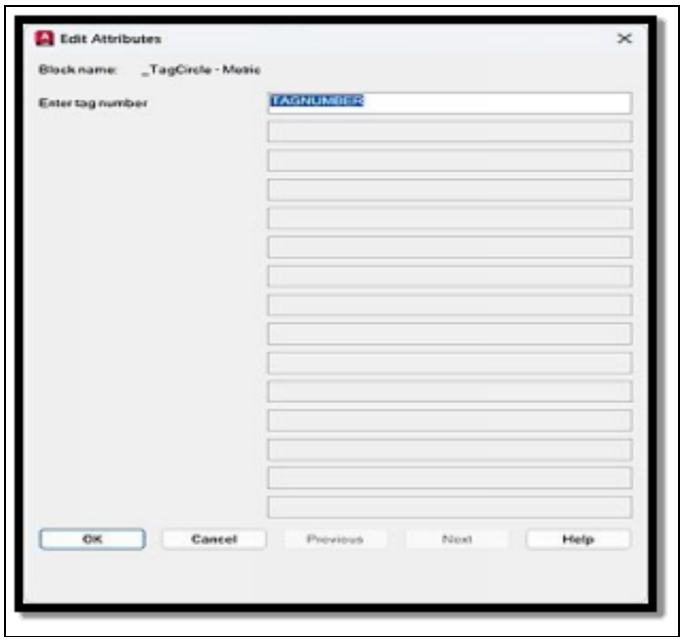
The leader is aligned with the circle, and a dynamic constraint is activated, visually connecting it to the crosshairs. AutoCAD will request the input of a second point:

**Indicate the desired location for the leader landing:**

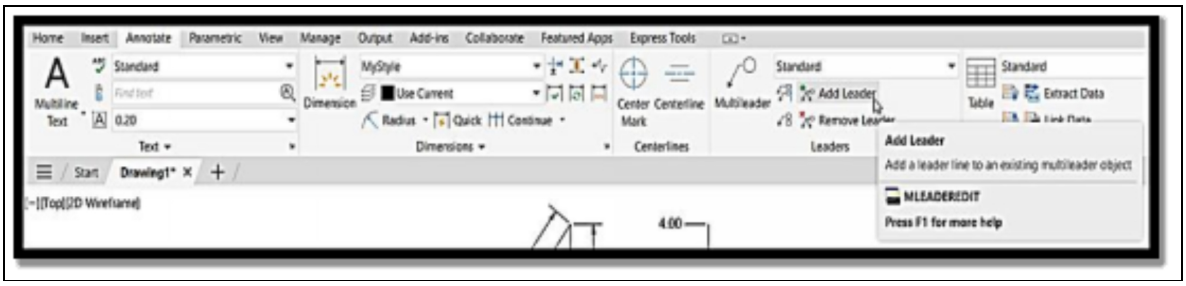
- Indicate the desired position for the leader, positioned at a 45° angle in an upward and rightward direction from the initial point.




When creating a drawing in AutoCAD, you can easily add a leader, landing line, and circle. Once you've done that, you can open the Edit Attributes dialog box, which will prompt you to enter a tag number. Attributes are essential components that store and link information to blocks within a drawing. These data can take various forms, like a part number or name, and can be easily accessed from the drawing database.




- Enter the value of 5 and then select the OK button. The number 5 does not hold any specific importance in this context. This is for demonstration purposes only. The number is included in the circle tag. The leader styles can be similar to this one, with the only difference being the shape of the tag, such as a box, hexagon, triangle, slot, or detail bubble. Then, you can easily add another multileader using the same annotation tag, which will connect to the other circle.
- To add a leader, go to the Leaders panel on the Annotate tab and click on the **Add Leader tool**.

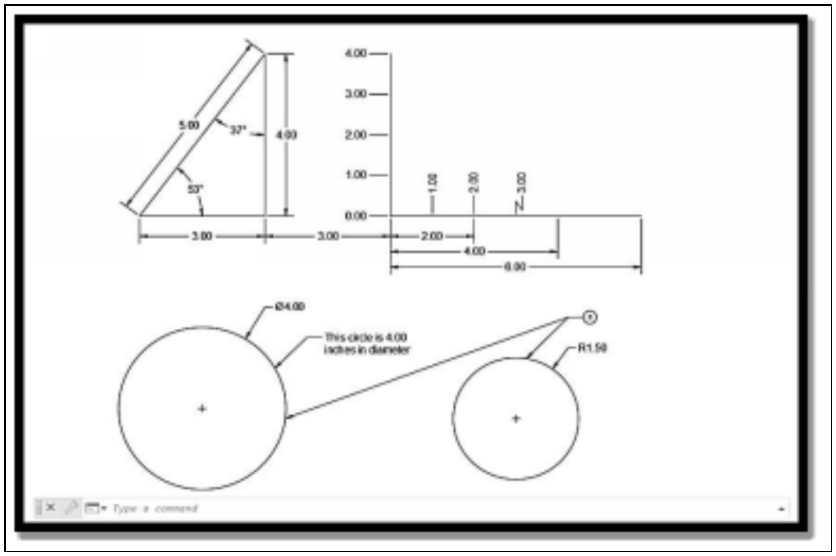



**AutoCAD provides prompts:**
**Choose multileader**

- Choose the multileader object that has the tag "5". AutoCAD seamlessly links a dynamic leader to the initial landing line,

allowing you to effortlessly extend it to a desired location.

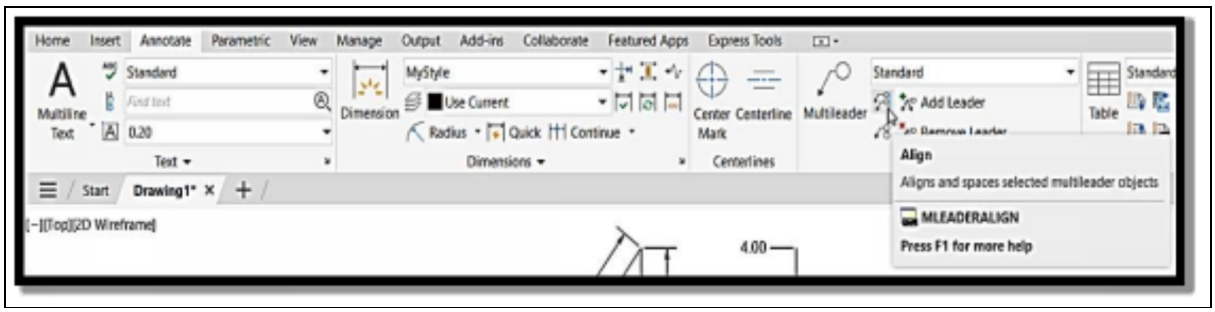
- Use the Nearest object snap to precisely align the added leader line with the 2.00 radius circle. AutoCAD offers the option to add another leader line to this multileader group, allowing for seamless continuation of leader lines.
- Press the **Enter key** to complete the command.




To align the radius dimension on the smaller circle with the multileader annotation tag, you can use a specific multileader alignment command. To achieve this, you will need to replace the radius dimension with a multileader object.

- Remove the dimension of **R1.50** from the smaller circle.
- To access the Multileader tool, navigate to the **Leaders panel** on the **Annotate tab** and click on it.
- To activate the **Nearest Object Snap,** simply use the <**Shift**> key and right-click. Then, choose the **Nearest option** from the Object Snap shortcut menu.
- Indicate a point on the circle close to where the previous radius dimension was connected.
- Choose a suitable spot for the leader landing to ensure that the leader text is placed near the previous dimension's location.
- Enter **R1.50** and press the **Enter key**.
- To finish editing the text, simply click outside the in-place text editor.

- To access the Align tool, navigate to the Leaders panel on the Annotate tab.

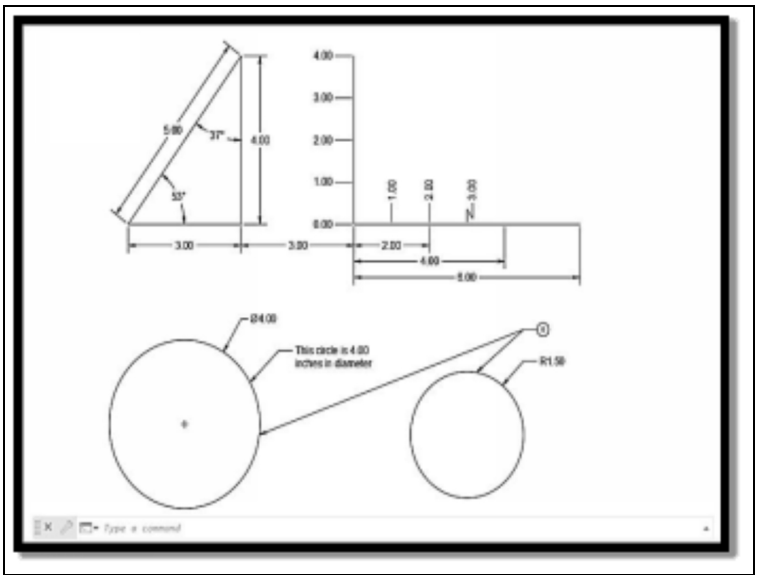



**AutoCAD prompts:**
**Choose the multileader:**

- Choose the multileader labeled as 5 and the R1.50 multileader.
- To conclude the selection of objects, simply perform a right-click.

AutoCAD prompts:
Choose the multileader you want to align to or select from the available options:
To achieve proper alignment, ensure that the upper tag is in line with the lower text.

- Choose the multileader below that contains the text "**R1.50**". There is an alignment line that can be easily adjusted to align at any desired angle. To achieve precise vertical alignment, enable **Ortho mode**.
- To enable Ortho mode, you can either click the **Ortho Mode button** located on the status bar or press the F8 key.
- To create a vertical alignment, you can specify a point at a 90° angle.




# How to Change Dimension Text

Editing dimensions in AutoCAD follow similar procedures as editing other objects. These elements can be manipulated in various ways, such as moving, copying, stretching, rotating, trimming, extending, and more. There are multiple methods available for modifying the text and placement of dimensions. Using grips is a highly efficient method for achieving various adjustments in object positioning. In this tutorial, you will discover two distinct methods for modifying dimension text.

# Using the DIMEDIT command

To modify dimensions in AutoCAD, simply enter the command "**dimedit**" and press **Enter**. When using AutoCAD, the software will present you with a series of options to choose from.
Enter the type of dimension editing [Home/New/Rotate/Oblique]
<Home>:
If dynamic input is enabled, you will also have the option to view these choices in a list format. Placement and orientation options in AutoCAD include Home, Rotate, and Oblique. The term "**New**" is used to indicate new text content.

- Choose the "**New**" option from the dynamic input list. When you launch AutoCAD, the Text Editor contextual tab will appear on the ribbon, ready for you to input your text. There is a small blue

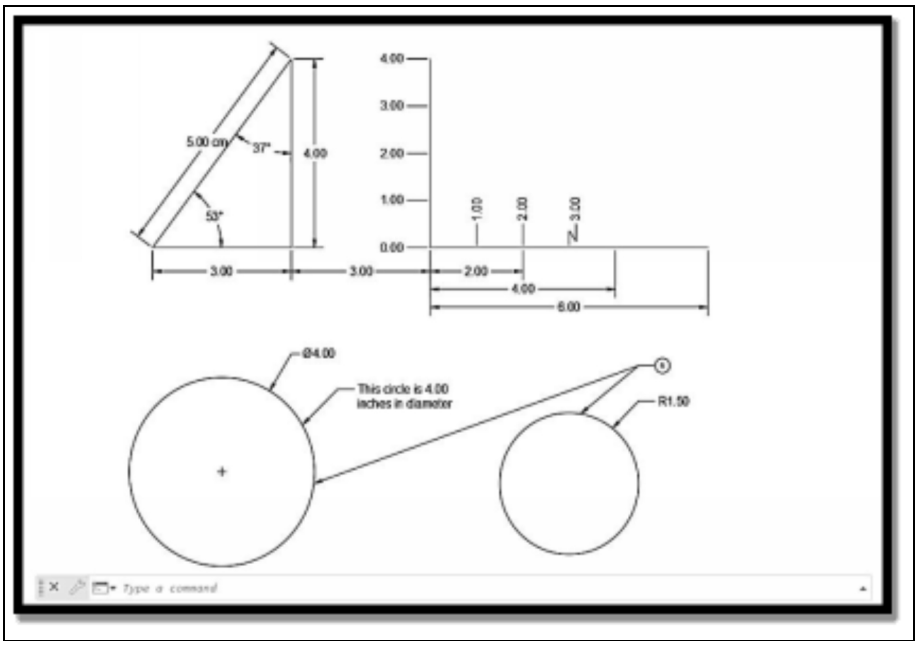
edit box that appears in the center of the drawing area. This box serves as the in-place text editor. To apply text to specific dimensions, simply enter the desired text and indicate the dimensions you want to apply it to. Modify the aligned dimension to display 5.00 cm.

- Remove the text within the in-place text editor.
- Enter the value as **5.00 cm.** The text is displayed in the in-place text editor.
- To close the in-place text editor and go back to the drawing area, simply click anywhere outside of it.

When using AutoCAD, the software will prompt you to select the objects that you want to receive the new text.

**Choose Objects:**

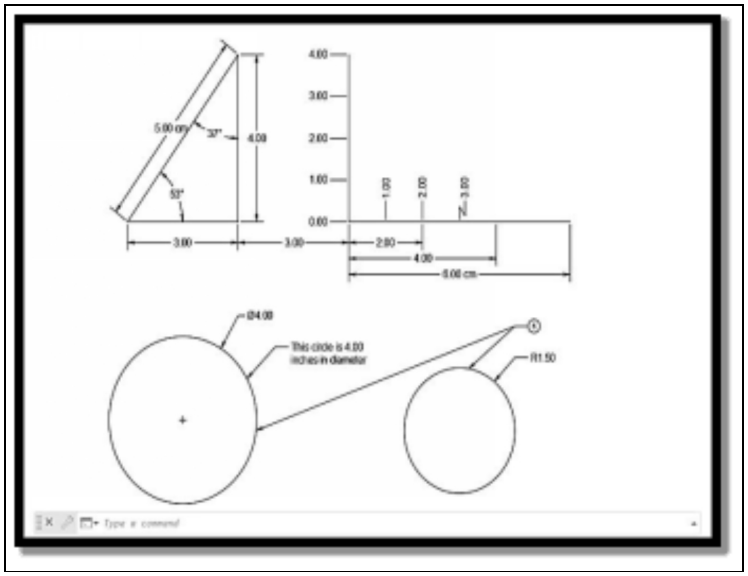
- Choose the aligned dimension of 5.00 on the hypotenuse of the triangle.
- Press the <**Enter**> key or right-click to complete the selection of objects.



# The Quick Properties and Properties Palettes

**Use the Quick Properties and Properties palettes to efficiently modify the content of dimension text. Here's what you can do:**

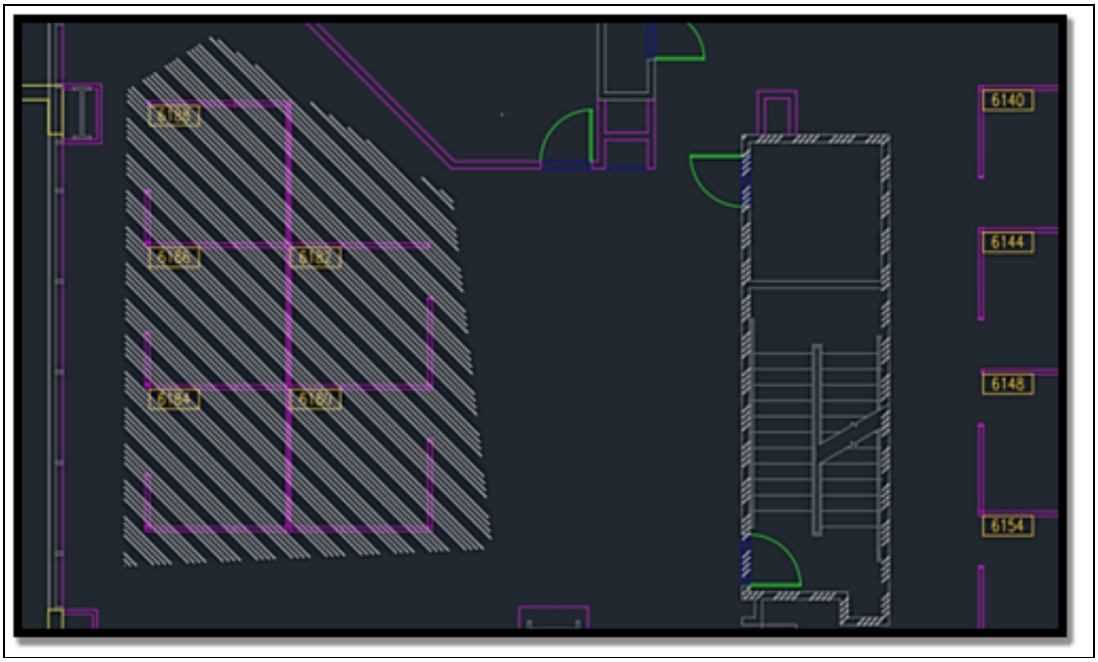
- Choose the baseline dimension below the 6.00 line.
- To access the Quick Properties feature, you can either right-click in the drawing area and choose Quick Properties from the context menu or simply click on the Quick Properties tool located on the status bar. (If needed, you can access Quick Properties by selecting it from the Customization menu located on the status bar.)
- Access the Text override field by clicking on it.
- Enter the value of 6.00 cm in the edit box.
- To close the Quick Properties palette, simply click on the "X" button located at the top right corner of the palette.
- Use the <**Esc**> key to easily clear grips.



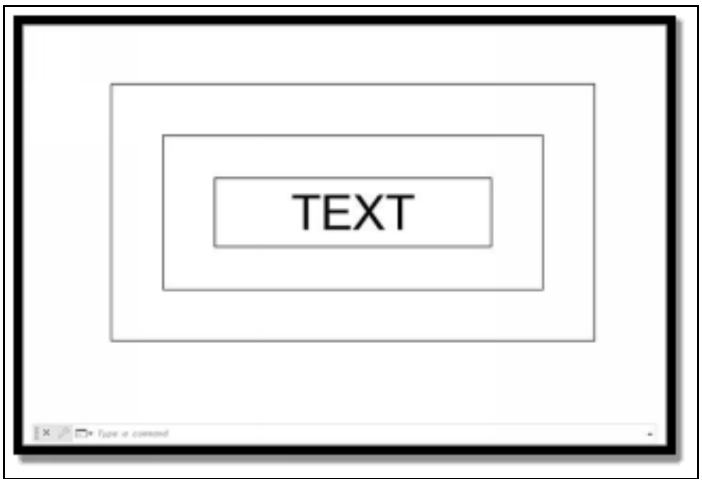

**Here's a helpful tip:** Aside from simply replacing a dimension value with static text, there is a way to include text before or after a dimension value. This can be done using the New option of the **DIMEDIT** command or by using the Text Override field found in the Quick Properties/Properties palettes. The characters <> in the text string represent the dimension value.

For instance, you can input <> cm to append cm after the dimension value in the previous exercise.

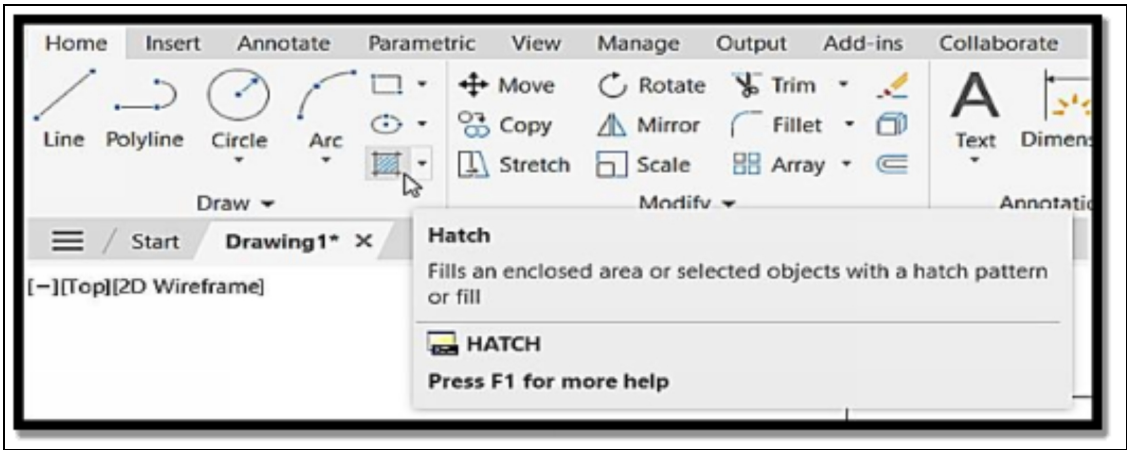
# How to Use the HATCH Command




Automated hatching is a fantastic feature that can greatly save you time. There are two fundamental methods for hatching in AutoCAD. One technique involves choosing a point within a designated area for hatching, and AutoCAD will automatically locate the closest boundary surrounding that point. When using an alternative approach, you can define the boundaries by selecting the desired objects. AutoCAD automatically generates associated hatch patterns as the default setting. Hatching in AutoCAD is dynamically associated with its boundaries, meaning that when the boundaries are modified, the hatching automatically adjusts accordingly. Hatching that is nonassociated is completely independent of the geometry it is placed within. Before starting this exercise, make sure to remove any existing objects from your drawing and switch to a nondimensional layer. Next, create three rectangles, each one nested inside the other. In the center of these rectangles, add the word "**TEXT**" with a height of 1.00. To achieve this, ensure that the height of your inner rectangle is at least 1.50 units.




- To access the Hatch command, navigate to the Home tab on the ribbon. From there, select the **Draw panel** and click on the **Hatch tool**.




When you initiate the HATCH command, you will see the Hatch Creation contextual tab. The ribbon tab provides a wide range of tools for creating hatch and fill patterns. You have access to 50 standard predefined patterns, solid color fills, and gradient fills, giving you plenty of options to choose from. The default pattern in AutoCAD is ANSI31, which is a straightforward pattern consisting of crosshatch lines at a 45° angle. If the pattern is not visible and highlighted on the Pattern panel, you may need to scroll to locate it.

- To navigate through the list of patterns, use the up and down arrows located on the right side of the panel. Since the list is organized alphabetically, except Solid, you can try scrolling upwards in the list.
- If needed, choose the **ANSI31 pattern image**.

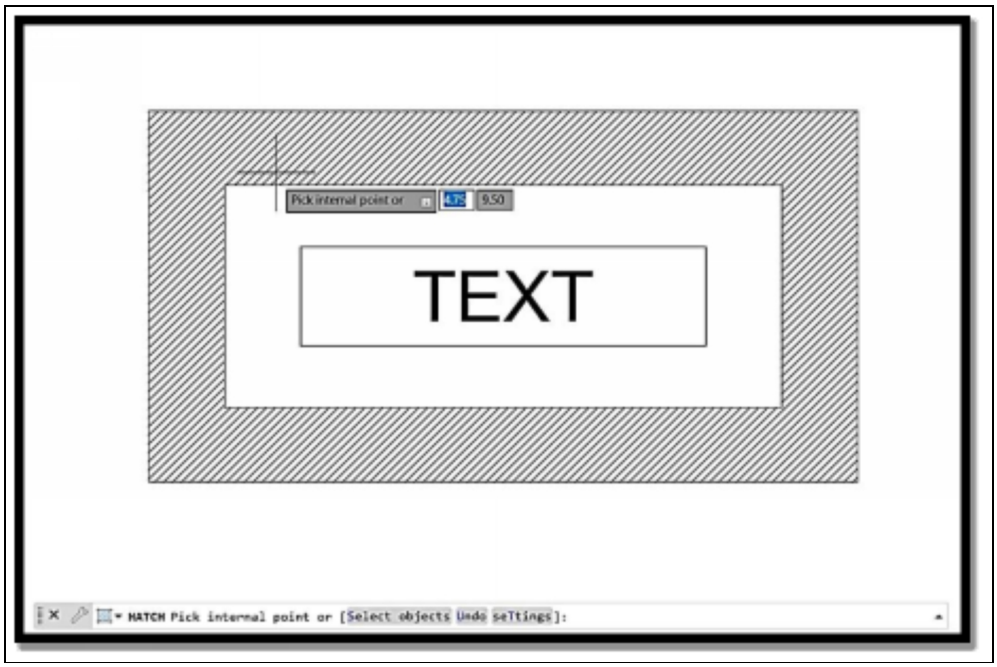
# AutoCAD prompts

**Pick internal points or [Select objects/Undo/seTtings]:**
This indicates that the default method being used is to select internal points. Pressing the S key will activate the select boundary objects method while pressing the T key will bring up the Hatch and Gradient dialog box. When the cursor is placed within a defined area, AutoCAD will automatically analyze the drawing to determine if there is a closed area under the crosshairs. If a closed area is detected, AutoCAD will then display a preview of the area with the appropriate hatch or fill pattern applied. For this particular scenario, the appropriate designation would be **ANSI31**.

# Note

The HATCH command in AutoCAD retains the most recent settings for your convenience. When selecting objects to draw a hatch pattern, it is possible for the default setting to be "**Select objects**" instead of using internal points. To accomplish this, you can easily choose **internal points** by using the command line.

- Position the crosshairs within the space between the largest and second-largest rectangles and hold.

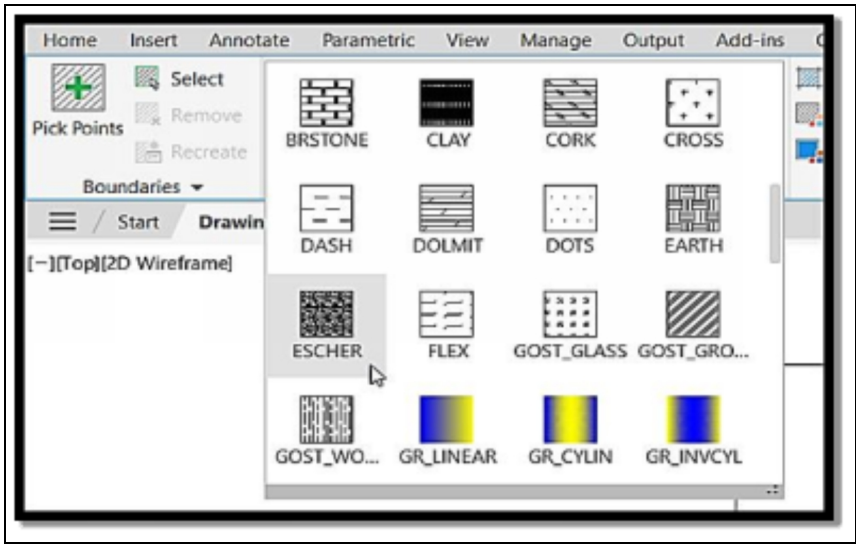



- Position the crosshairs within the space between the second largest and smallest rectangle, then hold your position. The preview displays the area with a hatched pattern. It is important to note that AutoCAD can identify both internal and external boundaries in each case.
- Position the cursor within the inner rectangle and hold. Observe in this preview image how AutoCAD effortlessly detects text within the boundary and intuitively creates space around it.
- Position the crosshairs outside of all the rectangles. You are currently in the HATCH command, however, no crosshatching has been applied to your drawing at this point.

# Additional Predefined Patterns

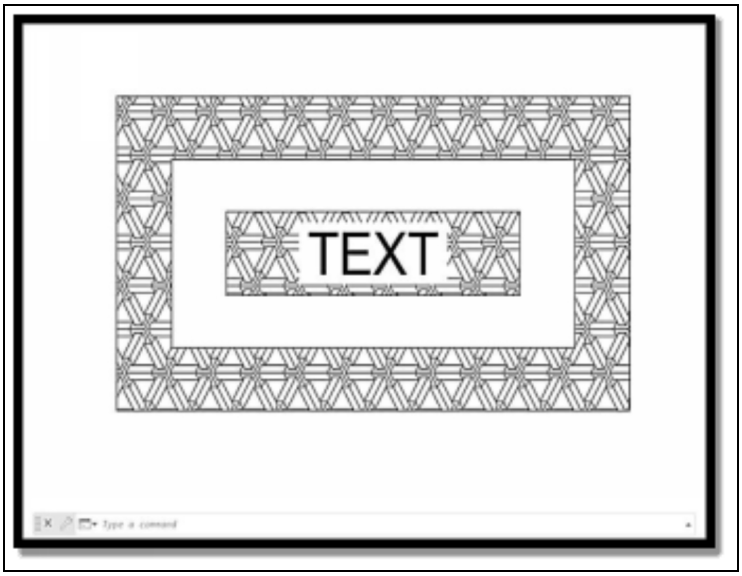
The Pattern panel in AutoCAD provides access to a comprehensive library of 82 predefined hatch patterns. To navigate through the pattern images, you can use the arrows located on the right side of the panel. To access a more expansive view, you can use the lower arrow. This will open a larger window featuring four rows of images, complete with its scroll bar on the right-hand side. To access the Escher pattern, simply open the window and scroll down.

- Locate the downward-facing arrow located at the lower right corner of the Pattern panel. Take note that this arrow is located at the bottom, distinct from the middle arrow. The bottom arrow also points downwards and serves the purpose of scrolling through rows of four patterns at a time.




- Locate the Escher pattern and choose it from the options below.

The window is closed, and the row displaying the Escher pattern appears on the Pattern panel with Escher selected. Apply the pattern to both the inner and outer rectangles.



- Position the cursor within the space bounded by the largest and second-largest rectangles, then indicate a point within this enclosed region.

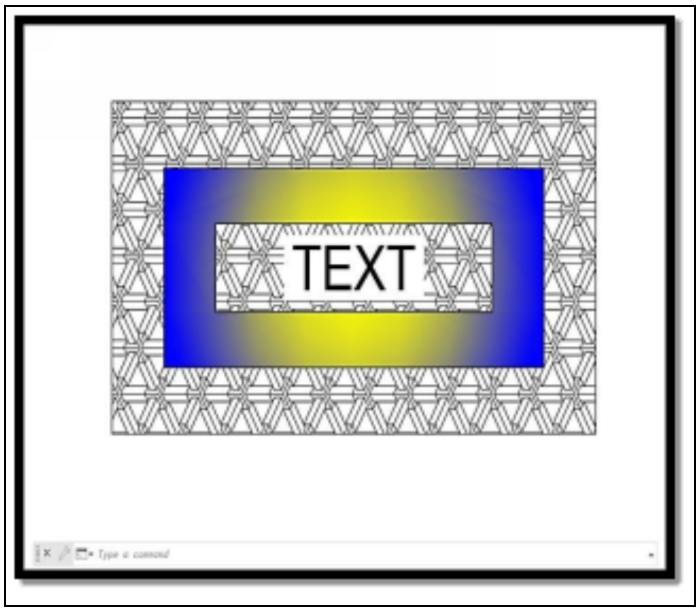
The area is filled with a complex and intricate pattern reminiscent of the works of M.C. Escher. To hatch the inner rectangle, you will need to exit and then re-enter the HATCH command. It is essential to consider that the boundaries of the inner rectangle fall within the boundaries of the outer rectangle. If you want to hatch separate areas, you can proceed without exiting the command.

- Press the Enter key to execute the command.
- Press the <**Enter**> key once more to reenter the HATCH command.
- Position the crosshairs within the inner rectangle, indicate a point within the enclosed area, and press <**Enter**> to finalize the command.

**Here's a helpful tip:**
For optimal results, it is recommended to first draw the text before hatching an area that includes it. When there is text in the drawing, AutoCAD will automatically create space around it when applying hatching. In this case, it is necessary to create a temporary boundary around the text, trim it to fit within the boundary, and then remove it completely.

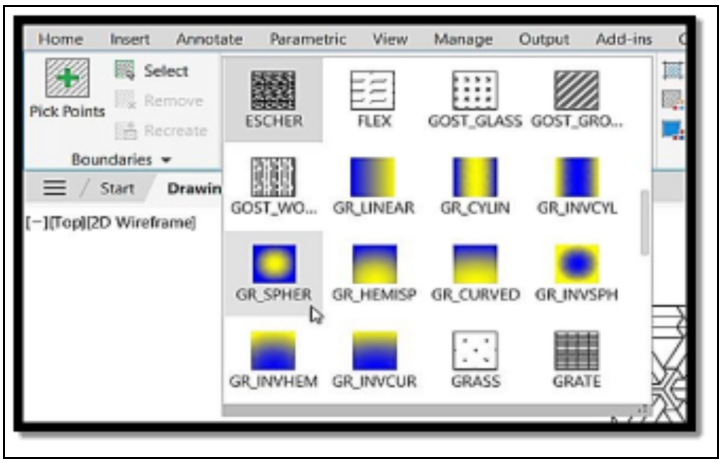
# Gradient Fill

Afterward, a gradient fill pattern can be applied to the space between the inner and outer rectangles. To select a predefined hatch pattern, follow the same process as choosing any other pattern. Alternatively, you can use the drop-down list located at the top left of the Properties panel on the **Hatch Creation contextual tab**. To use the previous technique, access the Pattern window and navigate to the desired gradient fill. To access the gradient patterns in the Patterns panel, simply choose the Hatch Type drop-down list on the Properties panel and select **Gradient**. This will take you directly to the desired options.

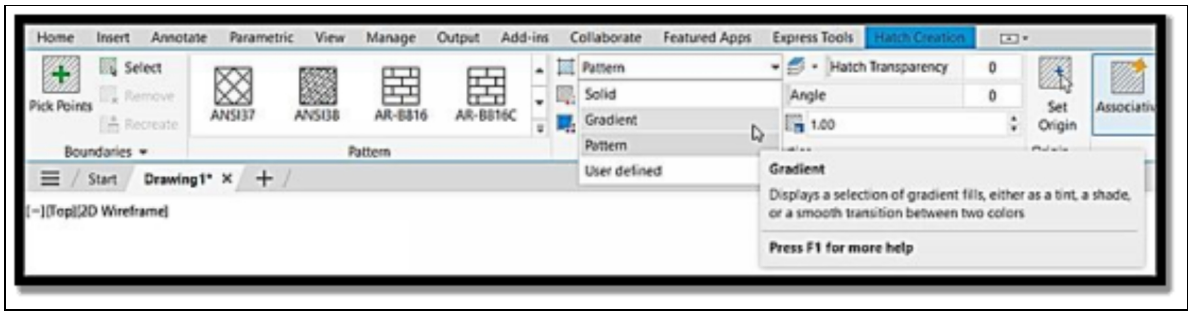



In AutoCAD, a gradient refers to a fill pattern that showcases a seamless transition between different colors or tints.

- Press the **Enter key** to access the **HATCH command** again. By selecting this option, you will be directed to the Hatch Creation contextual tab.




- Access the **Hatch Type drop-down menu** located at the top of the Properties panel.




**The drop-down list provides access to a wide range of solid fills in different colors, as well as user-defined hatch patterns. It also allows you to modify the colors in gradient fills.**

- Choose the **Gradient option** from the available list.
- Scroll down on the Pattern panel by using the arrow located at the bottom right.
- Choose the **GR_SPHER gradient pattern**, which can be found in the first pattern of the second row.
- Press the Enter key to exit the **HATCH command**.

# Scaling Dimensions Between Paper Space and Model Space

If you're familiar with creating multiple-viewport layouts, then this should be an old hat for you. When incorporating text and dimensions into your drawings, it's important to consider the scale relationships between model space and paper space. This can lead to questions and considerations that need to be addressed. With the Zoom XP feature, you can easily establish accurate scale relationships between model space viewports and paper space units. To streamline this procedure, you can leverage the Annotation property of annotation objects and styles to facilitate the alignment of dimension and text sizes with the corresponding zoom factors.

# Inserting text in paper space

- Specify the coordinate limits as (**0,0**) and (**12,9**). This is of utmost importance. Using different limits can make the exercise more challenging to follow.

- Create a circle with a diameter of **3.50 units**, perfectly centered at the coordinates (**3.00, 4.00**).
- Create another circle with a diameter of **2.50 units**, centered at the same point.
- Create a small circle with a diameter of **0.25** at the lower quadrant of the circle with a diameter of **2.50**.
- Create a rectangle by specifying the coordinates of its corners. For the first corner, use the coordinates (**6.50,2.75**), and for the second corner, use the coordinates (**10.00,5.25**).
- Open Layout1 and create a layout with D-size limits by using the **Page Setup Manager**. To access the Page Setup Manager tool, navigate to the **Output tab** and select the **Plot panel**. From there, you can make the necessary adjustments for your page setup.

Position three viewports. The lower-left viewport is defined by the coordinates (1,1) and (20,13). The coordinates for the lower-right viewport are (22,1) and (32,13). The viewport extends from coordinates (1,14) to (9,20). To optimize the display, set the zoom factor for the larger viewport to 2xp. For the smaller viewport on the right, set the position at 4xp and make sure it is centered on the midpoint of the left side of the rectangle. As for the smallest viewport at the top, position it at 6xp and ensure it is centered on the small circle located at the quadrant of the 2.50 circle.

# Text in Paper Space and Model Space

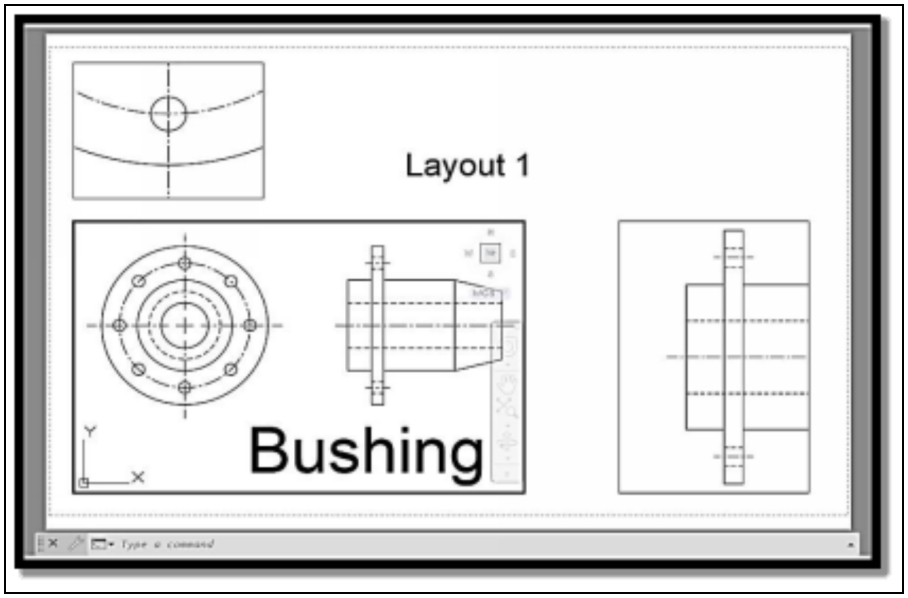
When working in paper space, it is possible to draw text at a 1:1 scale of the layout. When working with viewports, it's important to keep in mind that the scale of the viewport will have an impact on the text that is drawn within it.

**Here's a suggestion:**

- Ensure that you are in paper space by double-clicking anywhere outside the viewports.
- To access the Single Line Text command, navigate to the **Annotate tab** on the ribbon and click on the **Text option**.
- Choose a starting point that is located outside of any of the viewports, approximately at coordinates (**15,15**). The text is

positioned approximately 2.00 units above the right side of the left viewport.

- Enter "**1**" to set the text height to 1 unit. Assuming you are using the D-size paper space limits. If not, you will need to make adjustments to your settings. For A-size paper, the text height typically measures around **0.25**.
- Press the **Enter key** to rotate the object by 0 degrees.
- Enter "**Type Layout 1**" and press the **Enter key**.
- Press the <**Enter**> key once more to exit. It is important to include text in your drawing.
- To enter model space in the largest viewport, simply double-click in the lower-left corner of the viewport.
- Re-enter the **TEXT command**.
- Indicate the starting point below the left view in the viewport.



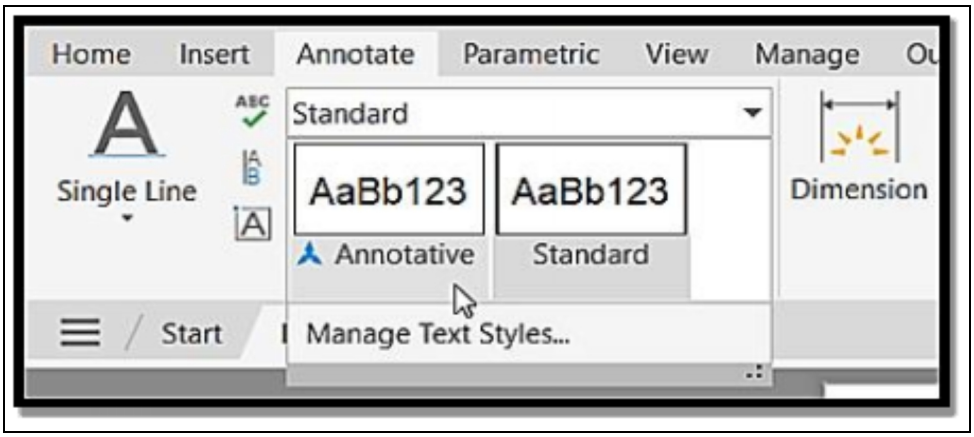

- Press the **Enter key** to keep the height at **1.00 units**. Also, press the **Enter key** to apply a rotation of 0 degrees.
- Enter "**Bushing**" and press the **Enter key**.
- Press the **Enter key** once more to exit the command. To create space for the text, it might be necessary to relocate objects within the viewport. To achieve the desired result, use the PAN function within the viewport to align your drawing.

What seems to be the issue in this situation? What is the reason for the discrepancy in size between "**Bushing**" and "**Layout 1**"? Can you recall the scale factor you used for the Zoom XP in this particular viewport? When the viewport is enlarged by two times in paper space, any text drawn inside it will also be enlarged by a factor of 2. Feel free to experiment with drawing text in the other two viewports. Text in the right viewport is magnified at a scale of four times the paper space size, while text in the uppermost viewport is magnified at a scale of six times. One way to address these enlargements is by dividing the text height by factors of 2, 4, and 6. However, this approach can quickly become quite cumbersome. In addition, if you choose to modify the zoom factor in the future, you will need to recreate any text that was drawn within the adjusted viewport. Otherwise, the text sizes throughout the drawing may vary, resulting in inconsistency. The annotative property is designed to provide greater control in situations like this.

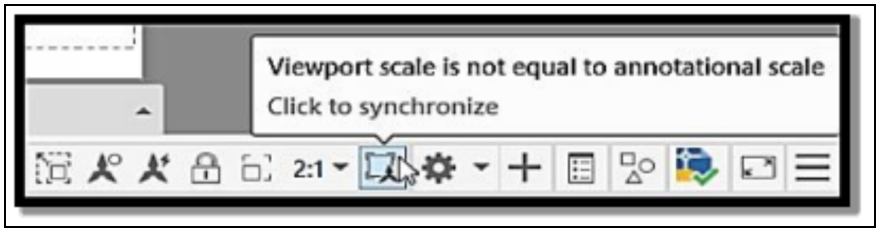
# The Annotative Property

Any objects used for annotating drawings can be classified as annotative. With the annotative property, you can easily attach multiple scales to the object. By ensuring the property and scale are set correctly, you can ensure that the object's scale aligns with the viewport's scale. This will result in the annotation appearing at the desired size in paper space. Various objects in AutoCAD can be assigned the annotative property. These objects encompass text, dimensions, hatch patterns, tolerances, leaders, symbols, and other explanatory symbols that can even be customized to your liking. You have the option to apply the property to specific annotative objects or include it in a style definition. First, you'll need to add the property that allows you to create text in the lower viewport, ensuring that it is at the correct scale.

- Remove the "**Bushing**" from the lower-left viewport.
- To access the Annotative Text Styles in AutoCAD, go to the Annotate tab, click on **Text**, and then select **Text Styles**.




Take a look at the right side of the status bar. In the model space in the lower-left viewport, a viewport scale indicator should be visible. Ensure that the viewport scale is set to 2:1, aligning it with the scale of the existing viewport. Adjacent to the Viewport Scale tool lies the Annotation Scale tool. When you hover your cursor over this tool, a tooltip will appear that states, "**Viewport scale is not equal to annotation scale: Click to synchronize**."

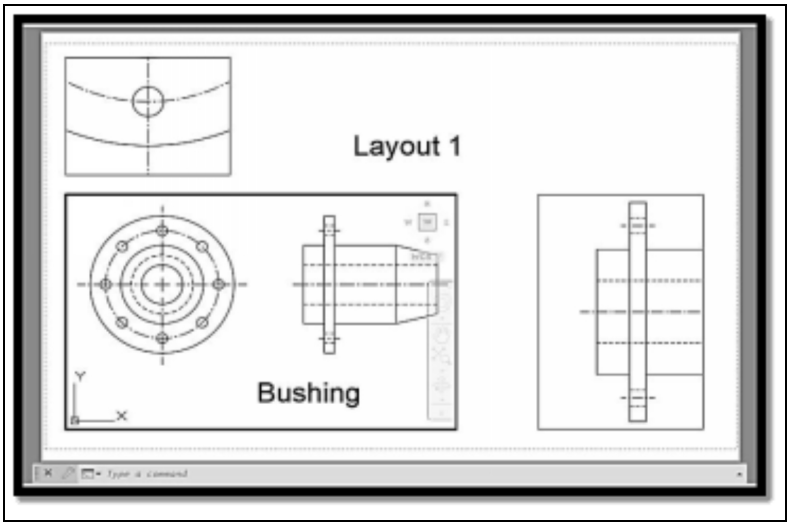



When you synchronize the annotation scale with the viewport scale, the annotative objects in the viewport will be automatically scaled. **This ensures that your annotations are properly sized and aligned with the rest of your drawing.**

- Click on the Viewport Scale Sync tool located on the status bar. It is important to observe that the viewport scale will adjust to 1:1. The scale factor in paper space is currently set to 1:1, which is the standard setting. AutoCAD automatically adjusts the viewport scale to align with the paper scale, ensuring synchronization. Observe the alteration in the viewport. To achieve a 2:1 ratio, you can adjust the settings accordingly.
- To change the viewport scale, locate the **Viewport Scale tool** on the status bar and choose the 2:1 option from the menu. The viewport

image reverts to its original size. Although the visual appearance may not appear different, the recent update ensures that the annotation and viewport scales are now synchronized. As a result, annotative text will automatically adjust to match the units used in paper space. Now, you can easily create text in the Annotative style within the left viewport.

- Select the **Single Line text tool** located in the Text panel of the Annotate tab on the ribbon.
- Indicate the desired starting point beneath the objects in the lower-left viewport.
- Enter "**1**" to specify the desired text height.
- Press the **Enter key** to rotate the object by 0 degrees.
- Enter "**Bushing**" and press the **Enter key**.
- Press the **Enter key** once more to exit the command.



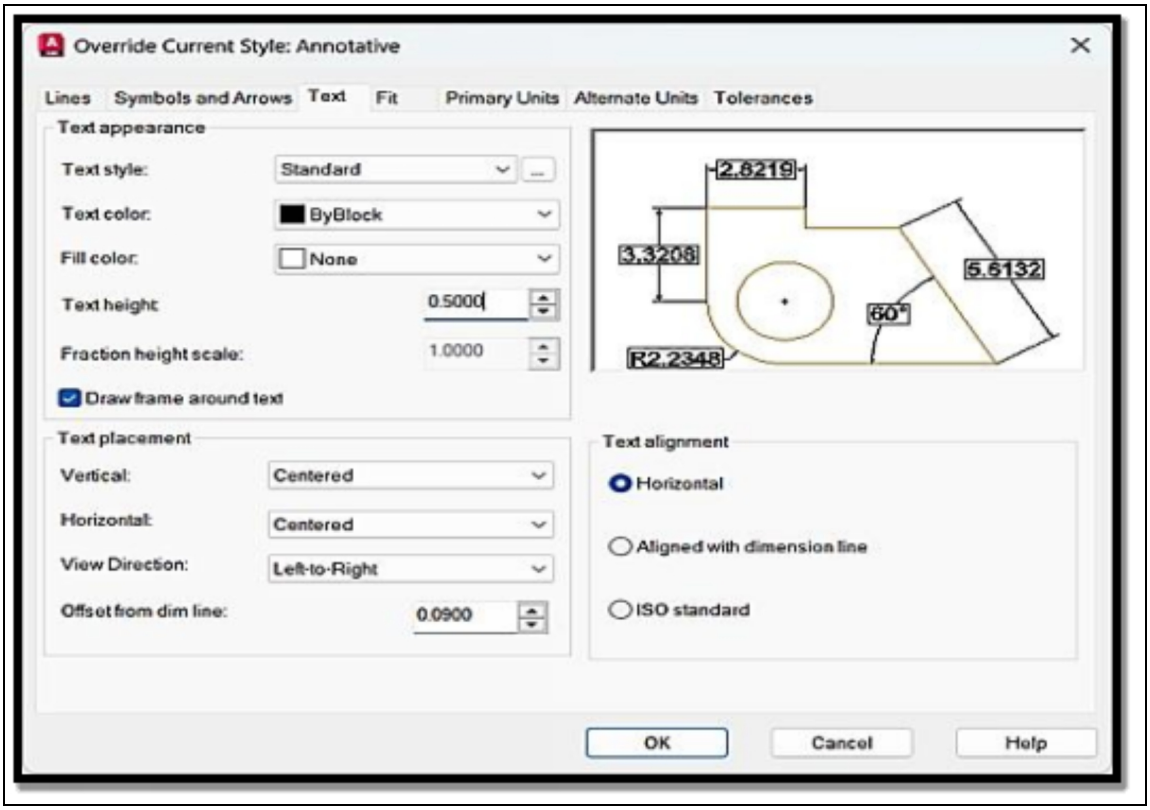

**Here's a helpful suggestion:** Once the scales of the viewports on a layout have been set, it is crucial to prevent any accidental changes to the scale within the viewport. To lock the scale of a viewport, simply select the desired viewports and use the **Viewport Lock tool** on the status bar or right-click and choose **Display Locked > Yes** from the shortcut menu.

# Controlling Annotation Scale Visibility

Afterward, you can incorporate dimensions into your viewports and gain a deeper understanding of managing scales and viewports. To begin, you'll want to switch over to the Annotative dimension style and then proceed to

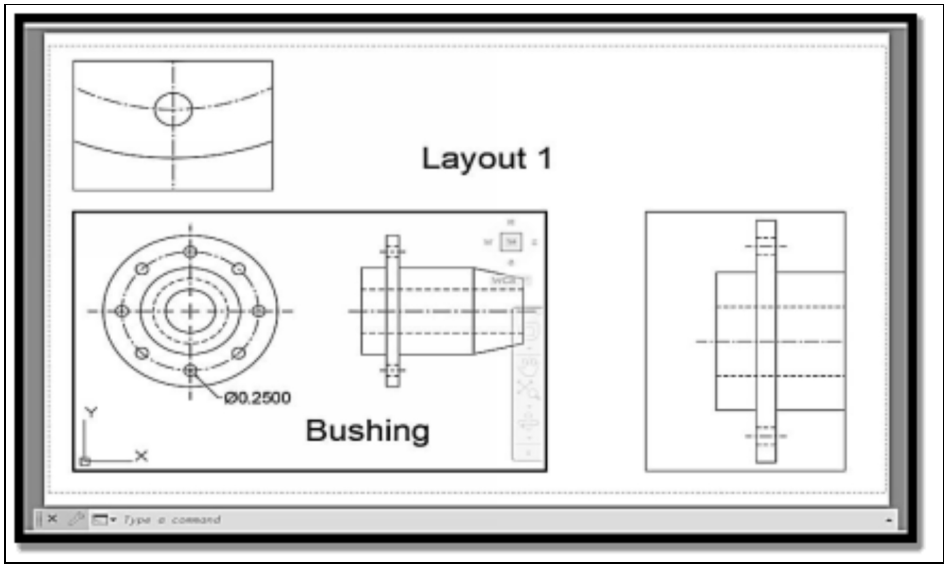
override the text within that particular style. **This is a temporary override and does not alter the default definition of the style.**

- To access the dimension style management in AutoCAD, go to the Annotate tab and select Dimensions. From there, choose **Dimension Style** and then **Manage Dimension Styles**.
- To set the Annotative style as the current one, simply double-click on it in the Styles list within the Dimension Style Manager. The label above the Styles list should now have the name "**Annotative**" added to it.
- Locate and select the **Override button** located on the right side.
- Click on the **Text tab** in the Override Current Style dialog box.



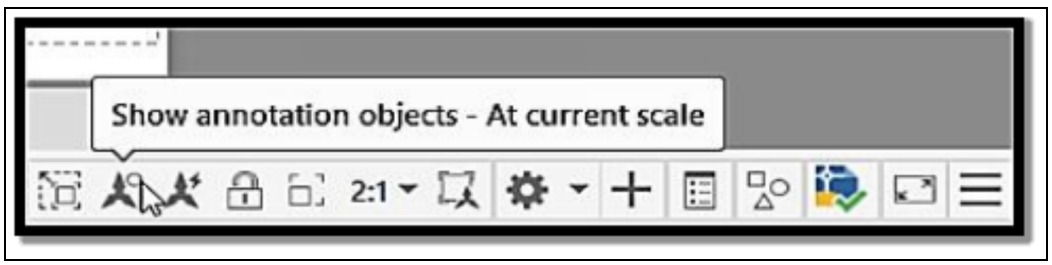

- Adjust the text height to 0.50. When correctly scaled, this will result in the dimension text being reduced to half the size of the 1.00 unit text that is currently displayed in paper space and the lower-left viewport.
- To exit the Override Current Style dialog box, simply click on the **OK button**.

- Close the **Dimension Style Manager**.
- To access the Diameter tool, navigate to the **Dimensions panel** on the ribbon's Annotate tab. Locate the Dimension tools drop-down menu and select the appropriate option.
- Select the bolt-hole circle located in the bottom quadrant of the bushing.




- Indicate the desired position for the dimension. It is important to note that the dimension recently added is not visible in the top viewport, even though the bolt hole being dimensioned can be seen in that viewport. The Annotative property of the text style has greatly facilitated this capability.

Take another look at the right side of the status bar. Located to the left of the Annotation Scale tool are four icons. This symbol in AutoCAD is annotated with a small gray circle positioned behind it. The tooltip for this tool will show the annotation objects at the current scale.



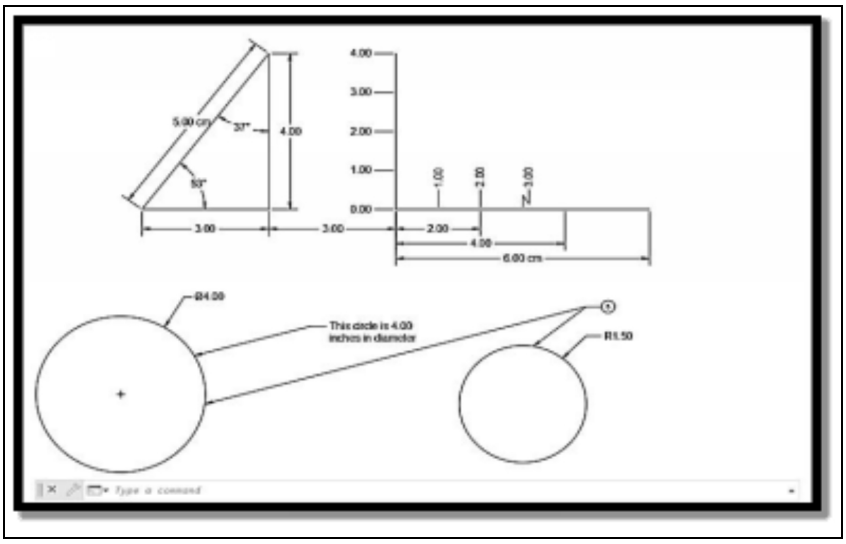
- Locate and select the **Annotation Visibility tool**. The color of the tool has been modified to blue. The tooltip now shows the option to display annotative objects at all times. Take a closer look at the viewport located in the upper-left corner. A dimension leader of significant size has been incorporated. When panning to view the dimension text, you will notice that it has a height of 1.50 units, which is three times the height of the text in the lower-left viewport, measuring 0.50 units. Here is the outcome of the viewport scales: the lower viewport is set at a ratio of 2:1, while the upper viewport is set at a ratio of 6:1.

# How to Use Associative Dimensions

Typically, dimensions in AutoCAD are automatically linked to the objects they measure, ensuring their association with the geometry. Changes made to the dimensioned objects are automatically updated in the dimensions. When an object with dimensions is relocated, the associated dimensions will also move accordingly. When an object is scaled, its position and measurements adjust accordingly to match the new size of the object. In this exercise, we will explore various aspects of associative dimensions. An associative dimension is a dimension that is linked to the object it measures. This means that if the object is modified or resized, the dimension automatically adjusts accordingly.

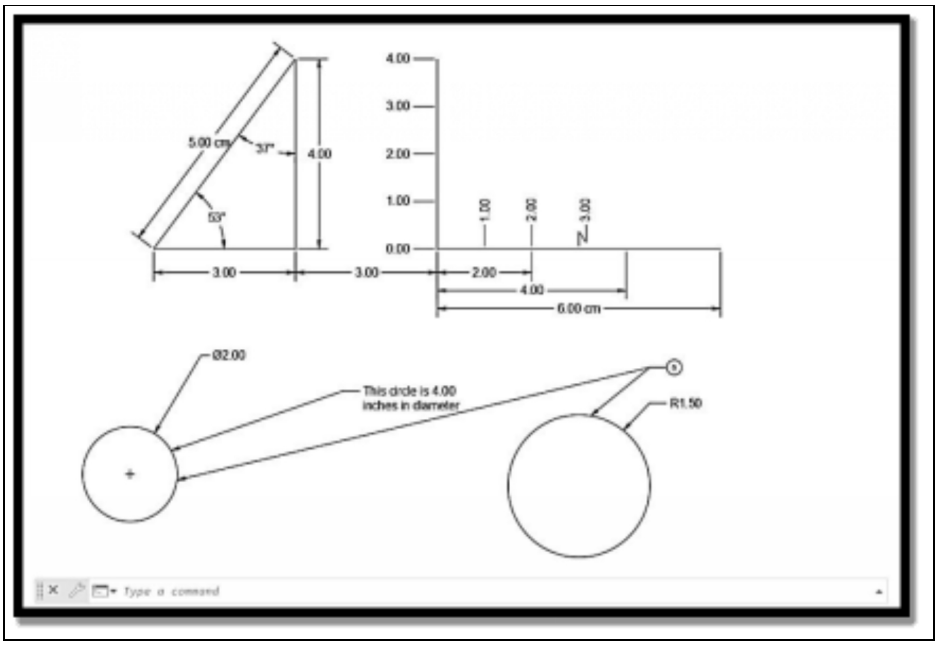
- Choose the circle with a radius of **2.00**.
- First, select the center grip of the circle. Then, effortlessly shift the circle approximately 2.00 units towards the left. Finally, click once more to finalize the move. It is important to observe that the dimension with a diameter of 4.00 moves along with the circle. Additionally, the leaders from the mtext dimension text and the multileader tag stretch to stay connected with the circle. However, the text and tag themselves do not move.

**To enhance your knowledge, let's begin by following these steps:**




- Choose one of the grips available.
- To access the Grip shortcut menu, simply right-click.
- Choose the appropriate scale.
- Enter a scale factor of 0.5 to adjust the size.
- Use the <**Esc**> key to easily clear grips.

It is important to observe that the scale factor is accurately represented in the diameter dimension, resulting in a change from 4.00 to 2.00. However, it is worth noting that the leadered text does not reflect this scale factor adjustment. The diameter dimension in AutoCAD is a dynamic and intelligent dimension that automatically adjusts and updates itself to accurately represent any changes made to the circle. The connection point of the leadered dimension is linked to the circle, while the text itself remains unassociated. Thus, the leader makes an effort to maintain a connection with the circle, while the text remains stationary and unaffected by the circle's movement.




# Changing Associativity of Dimensions

You can make nonassociative dimensions associative by using the **DIMREASSOCIATE** command. You can disassociate associated dimensions by using the **DIMDISASSOCIATE** command or by copying a dimension object. For each scenario, simply input the command and choose the dimensions to either link or unlink. **Here's an example for you to try:**

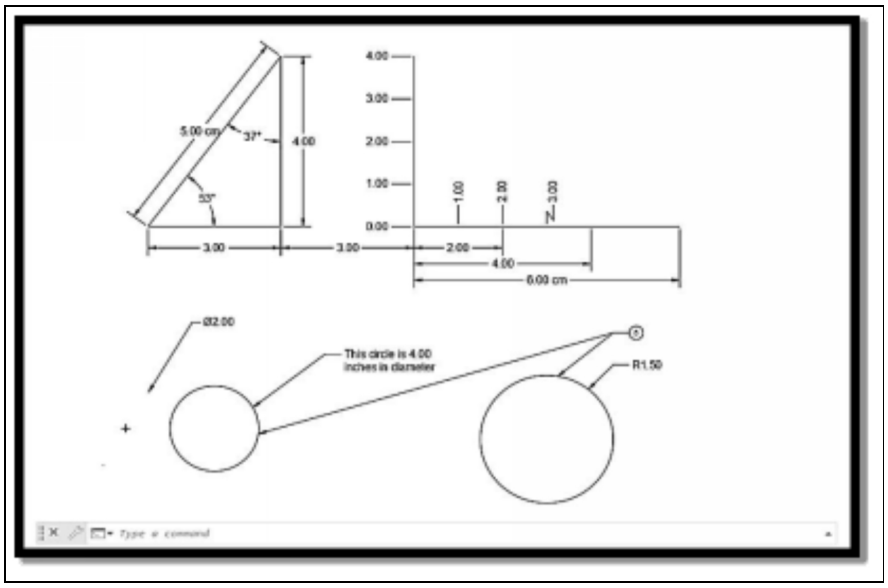
- Enter "**dimd**" and then choose "**DIMDISSOCIATE**" from the AutoComplete list. Dimdisassociate is not readily accessible from any ribbon panel or menu. However, this presents an excellent opportunity to leverage the AutoComplete feature.

**AutoCAD prompts:**
Choose dimensions to remove their association . . .
Choose objects:

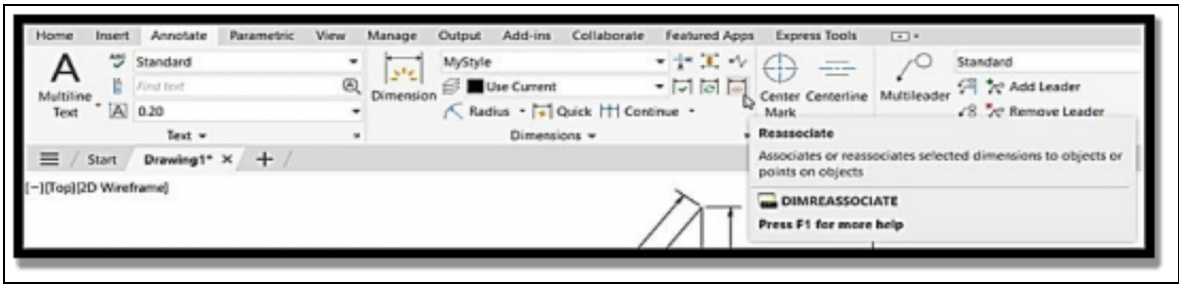
- Choose a dimension with a diameter of **2.00**.
- To conclude object selection, simply perform a right-click. Now, attempt to relocate the circle once more to observe the alterations.
- To move the circle to the right, simply select it and click on the center grip.

In this case, the diameter of 2.00 does not remain attached to the circle. The multileader will automatically adjust to remain attached to the circle, but the diameter dimension is not currently linked to the circle. To reassociate it, you can use the **DIMREASSOCIATE** command.

**Here's how:**



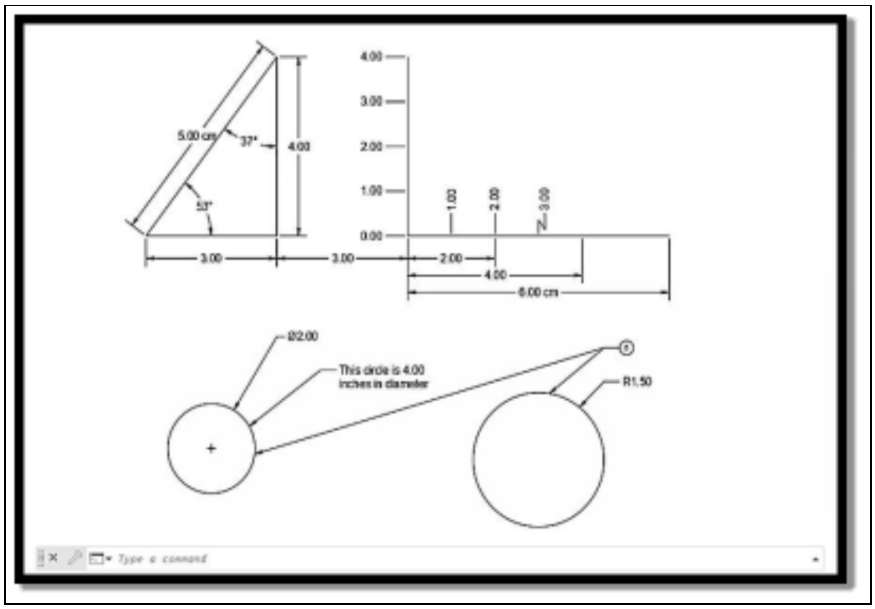

- To reassociate dimensions in AutoCAD, you have a couple of options. One way is to type "**dimr**" and select **DIMREASSOCIATE** from the AutoComplete list. Alternatively, you can go to the ribbon and select **Annotate** > **Dimensions > Reassociate**.




- Choose a dimension with a diameter of **2.00**.
- Press the <**Enter**> key to complete the selection of objects. AutoCAD identifies this as a diameter dimension and prompts for a circle or arc to link it with. It is important to note that this

allows you to link the dimension to any arc or circle, regardless of its previous attachment.

- Choose the circle on the left once more. The diameter dimension is repositioned and connected to the circle with a radius of 2.00.



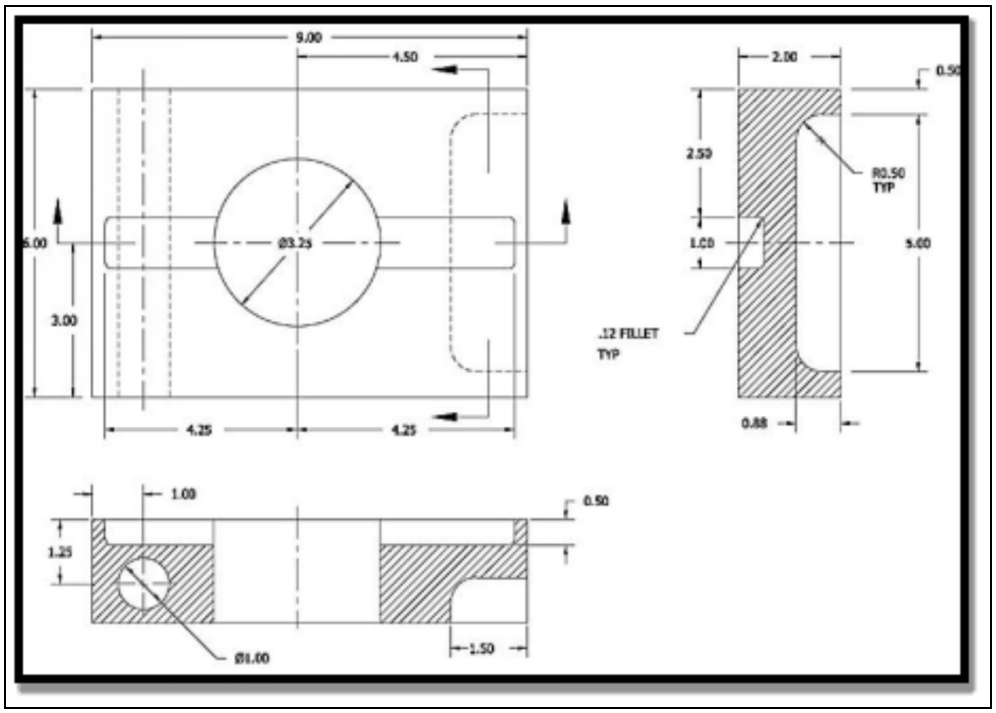
# Chapter Drawing Projects

# Drawing 8-1: Tool Block [INTERMEDIATE]

The dimensions in this drawing should function properly without any need for editing. The hatch is a commonly used pattern that is created by the user to indicate that the front and right views are sectioned views.

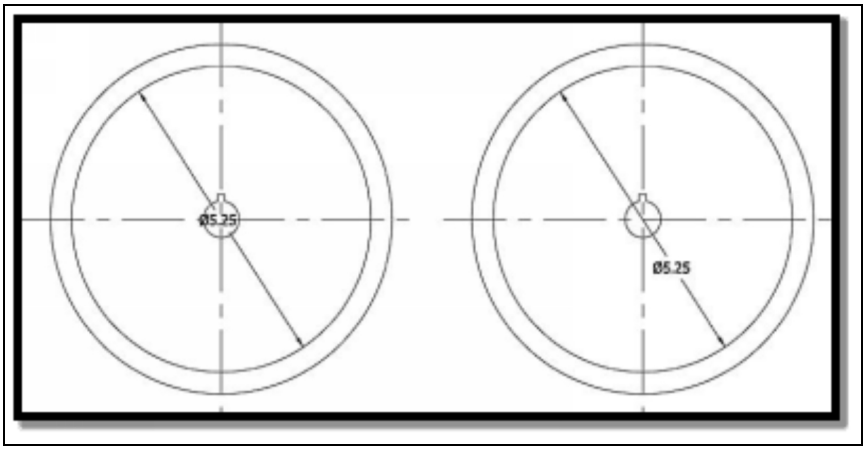
**Here are some suggestions for your drawings:**

Set the grid spacing to 1.0, disable the snap function, and adjust the hatch line spacing to 0.125. It is recommended to finish the drawing, including crosshatching, before adding dimensions and text at the end. Ensure that all hatching is placed on the designated hatch layer. After the hatching process is finished, make sure to assign it to the dim layer and disable the hatch layer. This will prevent any hatch lines from causing any disruptions when you need to select lines for dimensioning. Section lines in this drawing can be effortlessly created using leaders. First, adjust the dimension arrow size to 0.38. Ensure that Ortho is enabled before starting the leader. Begin the leader at the tip of the arrow and create a right angle, following the provided illustration. Once you have selected the second endpoint of the leader, simply press the <**Enter**> key to display the initial annotation prompt. To create a space, simply press the spacebar and then hit the **Enter key**. Press the <**Enter**> key once more to exit. To place the 3.25-diameter dimension at the center of the circle in the top view, make sure to set the **DIMTIX** variable to On. On the other hand, if you want to create the leader style diameter dimension in the front section, set the DIMTIX variable to Off.



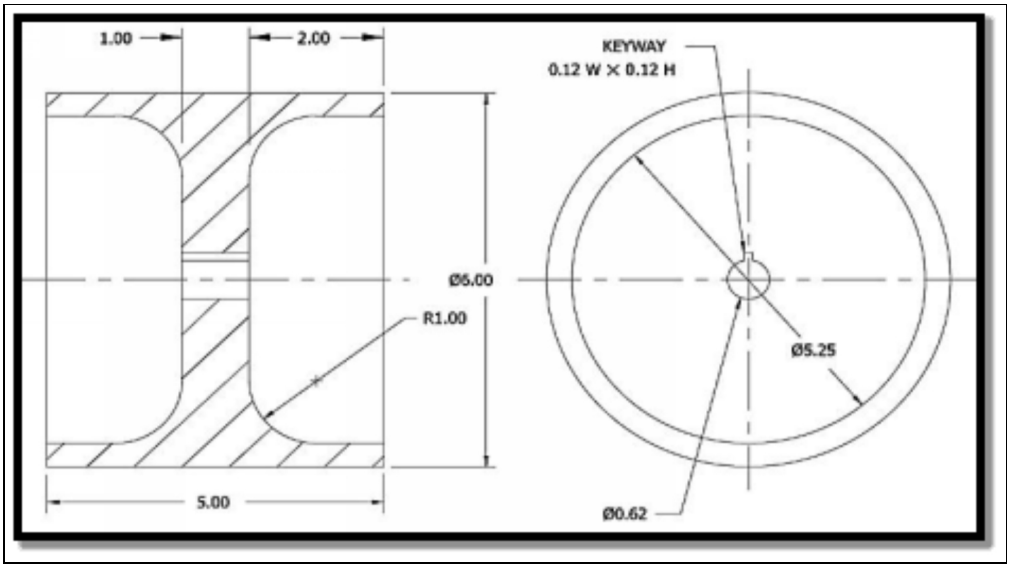

# Drawing 8-2: Flanged Wheel [INTERMEDIATE]

The objects in this drawing are quite simple and easy to understand. Creating a keyway can be effortlessly achieved by using the **TRIM command**. If needed, use the Edit shortcut menu or grips to relocate the diameter dimension, as demonstrated in the reference.




**Here are some suggestions for creating drawings:**
Set the grid spacing to 0.25, snap spacing to 0.0625, and hatch line spacing to 0.50.

- For drawing the keyway, it is recommended to use a snap value of 0.0625. Create a square with dimensions of **0.125 × 0.125** at the uppermost part of the circle with a diameter of **0.63.** Lower the vertical lines into the circle to enable them for circle trimming. To trim the circle and the vertical lines, you can use a window selection to choose both as cutting edges.
- Make sure to set the layer to hatch before applying the hatch, layer text before inserting any text, and layer dim before adding dimensions.




# Drawing 8-3: Knob [INTERMEDIATE]

This drawing uses the hatching and dimensioning techniques that were covered in the previous two drawings. Moreover, AutoCAD employs angular dimensions, leaders, and the "**c**" symbol for denoting diameter.

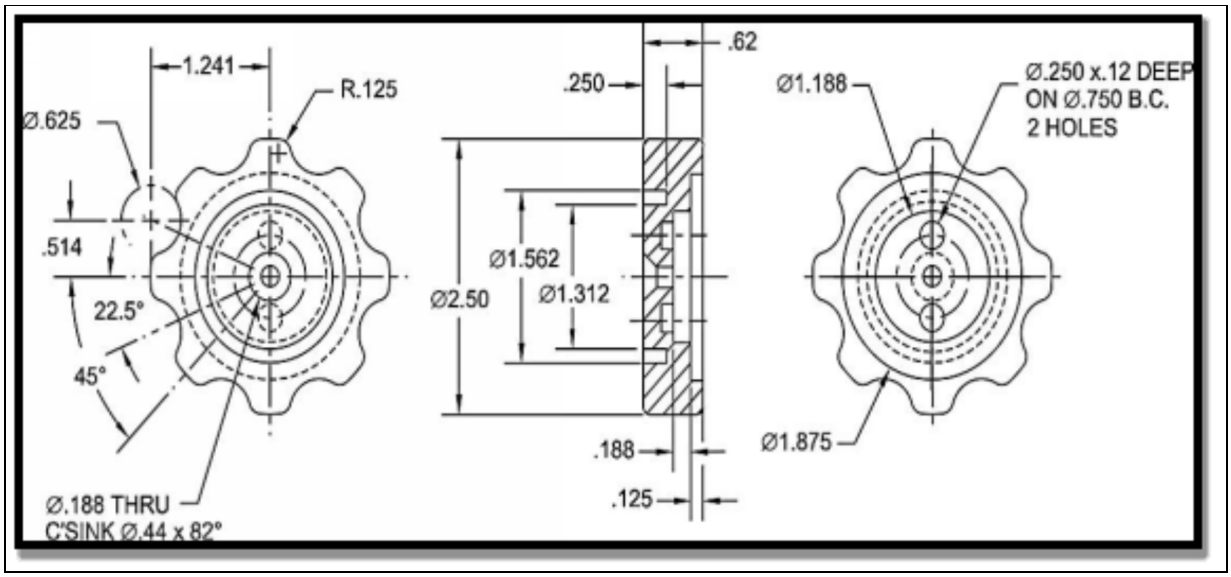
**Here are some suggestions for your drawings:**

Set the grid spacing to 0.125, snap spacing to 0.0625, and hatch line spacing to 0.25.

- Using the **MIRROR tool** can be a time-saving technique for creating the right-side view of this drawing. It's important to note that hatching should not be done before mirroring, as the **MIRROR command** will reverse the angle of the hatch lines.
- To achieve the indentations in the knob on the circular views, you can create a polar array using a circle with a diameter of

0.625. Make any necessary trims.

- Use the **DIMEDIT command** to add notes to the hole dimensions.
- It is important to observe that the diameter symbols in the vertical dimensions on the right-side view are not automatically generated. To add the diameter symbol to the text, use the %%c command.




# Drawing 8-4: Nose Adapter [INTERMEDIATE]

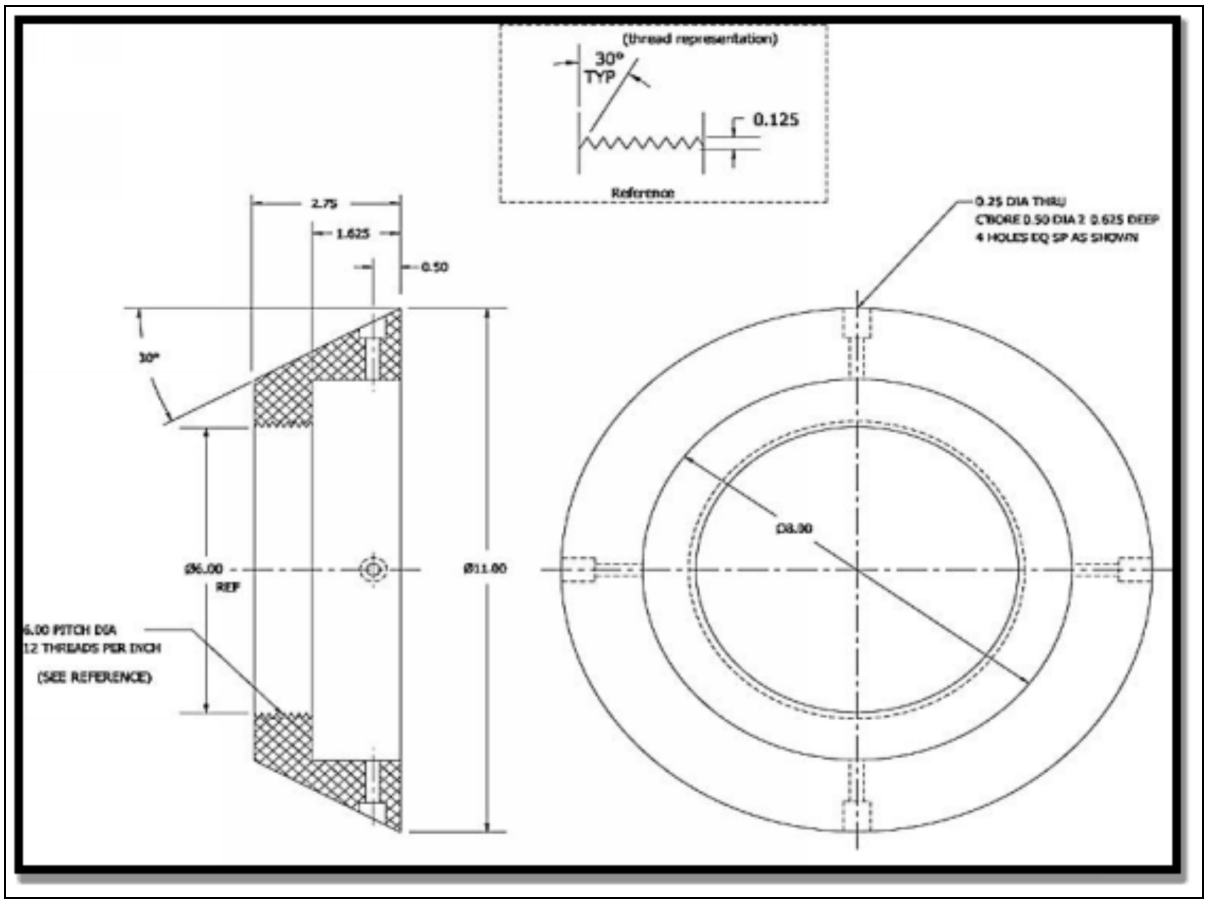
Use the ZOOM function extensively to focus on the intricate details of this drawing. Take note of the larger limits that have been set, which deviate from the usual settings. Additionally, the snap feature has been adjusted to be more precise in comparison.

**Here are some suggestions for your drawings:**

Set the limits of the drawing to (0,0) for the lower left corner and (36,24) for the upper right corner. Set the grid spacing to 0.25 units, the snap spacing to 0.125 units, and the hatch line spacing to 0.25 units.

- To accurately draw the thread representation shown in the reference, it is essential to use a snap value of 0.125. It is important to note that this is simply a standard representation for screw threads and does not provide actual dimensions. When you zoom in closely, drawing it should be a breeze.

- This drawing showcases two illustrations of simplified drafting techniques. There are two aspects to consider: the thread representation and the method of drawing counterbores in the front view. For an accurate representation of these holes, it is important to depict them as ellipses due to the angled nature of the object. Displaying these ellipses in the front view would result in a drawing that is perplexing and lacks practicality. It is recommended to use a simplified representation in such cases.




# Drawing 8-5: Plot Plan [INTERMEDIATE]

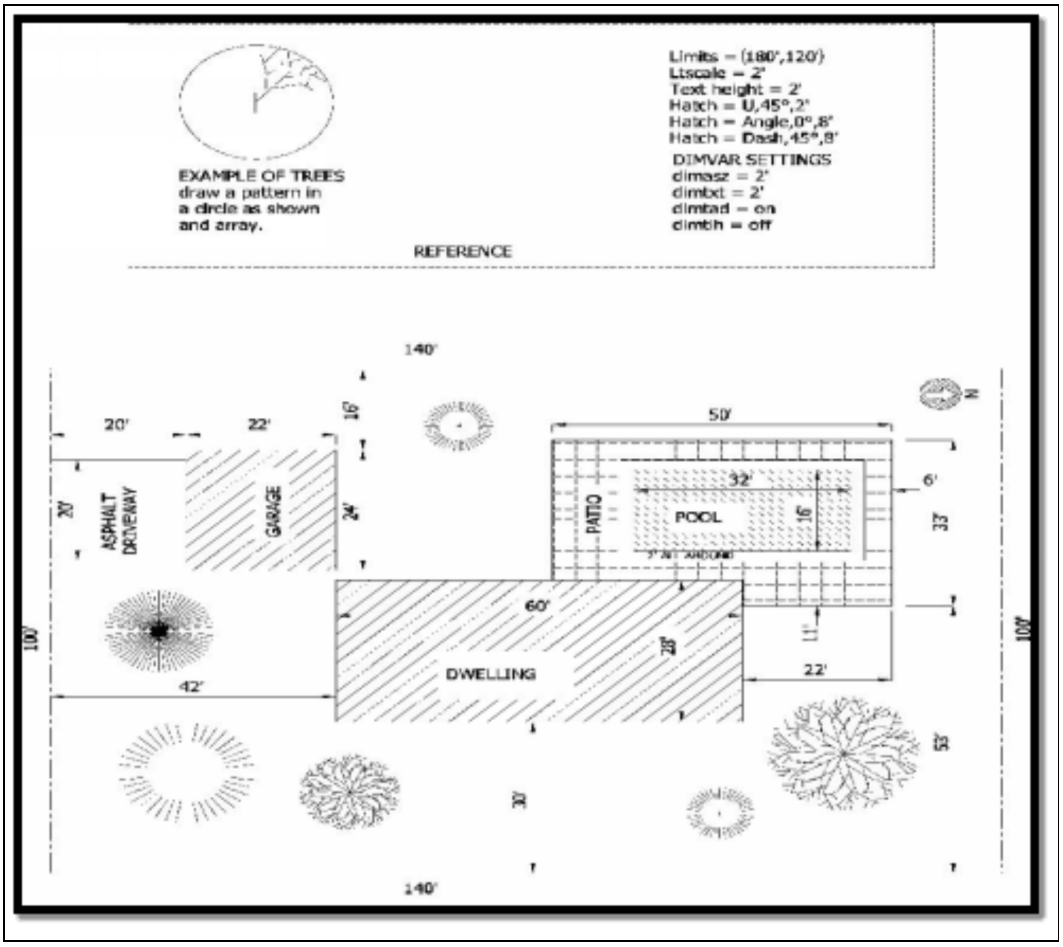
This architectural drawing uses three hatch patterns and incorporates multiple changes to dimension variables. Ensure that you configure these settings exactly as depicted.

**Here are some suggestions for creating drawings:**

The grid spacing is set to 10 feet. Set the SNAP value to 1 foot. Set the limits of the drawing area to (0',0') in the lower left corner and (180',120') in

the upper right corner. The LTSCALE command is used to set the line type scale in AutoCAD. In this case, the line type scale is set to 2 feet.

- The symbols depicted in this illustration represent various types of trees, including oaks, willows, and evergreens.
- When using **HATCH**, it is possible to create a defined boundary around text. However, there may be instances where you require additional white space beyond what HATCH provides. One effective approach is to create a rectangular shape that encompasses the text area, serving as an inner boundary. Afterward, you have the option to delete the box, resulting in a distinct area of space surrounding the text.




# Drawing 8-6: Panel [ADVANCED]

This drawing focuses on the application of ordinate dimensions. This powerful feature greatly enhances the process of drawing objects and

adding dimensions, making it much more efficient.

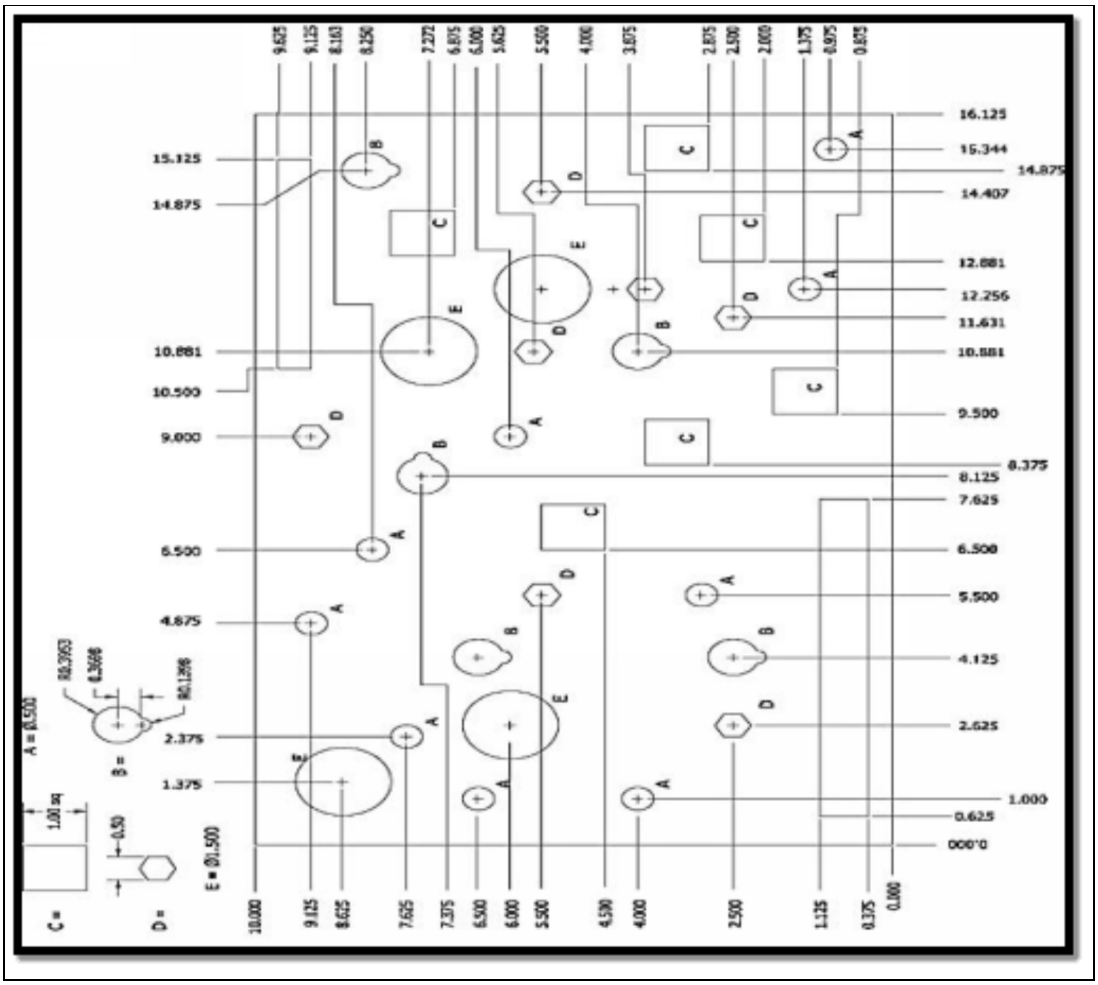
**Here are some suggestions for your drawings:**

Set the grid spacing to 0.50 units and the snap spacing to 0.125 units. The units should be set to three decimal places. Once the grid, snap, and units are set, you can proceed to create a new user coordinate system. Simply adjust the origin by moving it inward and upward by approximately 1 unit in each direction.

**Here is the procedure for your reference:**

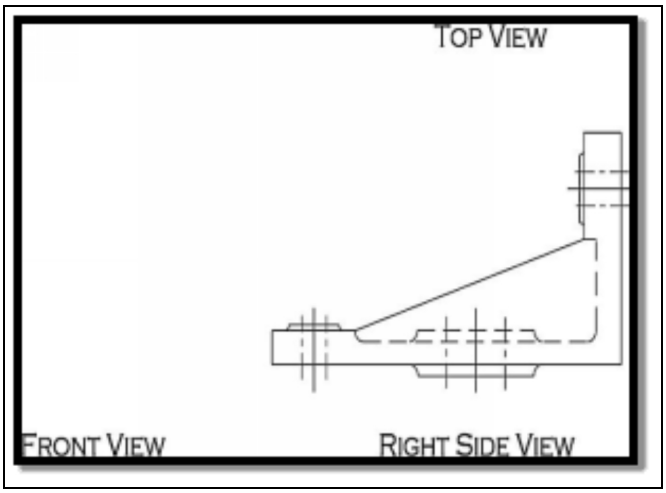
- To access the Origin option, go to the ribbon and click on View, then select Coordinates. From there, you can click on **Origin**. To make the Coordinates panel visible, you can right-click over the **View tab** and choose **Show Panels > Coordinates**.
- Select a new origin point.
- Moving forward, you can effortlessly position all the elements in the drawing by using the x and y displacement.

Here are some tips and tricks for using AutoCAD like a pro: After placing the objects, you can easily switch to the dim layer and start dimensioning using the ordinate dimension feature. To ensure precise alignment, it is important to maintain consistent dimensions on both sides of the panel as you navigate through the project efficiently. It is important to ensure that the leader endpoints align perfectly along a vertical or horizontal line.



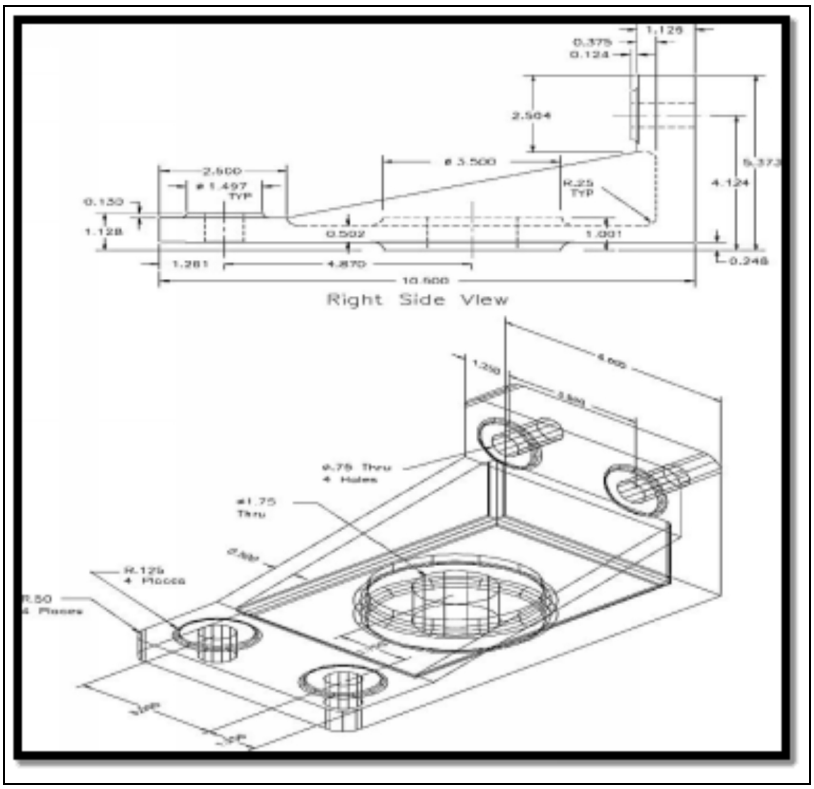
# Drawing 8-7: Angle Support [ADVANCED]

For this drawing, it is necessary to use the 3D view to generate three orthographic views. Create a front view, top view, and side view. The front and top views are accurately depicted, including all required hidden lines, while the right-side view is presented in a complete section. Ensure that the final multiview drawing is complete with all necessary dimensions.




**Here are some suggestions for your drawings:**

- Begin the drawing by establishing the top view. Use the top view to align the front and side views.
- To convert the hidden lines to solid lines, you can use the **HATCH command** to create crosshatching.
- Ensure that the right-side view is fully sectioned, using the **ANSI31 hatch pattern**.
- Make sure to include all the required hidden lines and centerlines in every view.


1.125
0.375
0.124
2.504
5.373
4.124
ø 3.500
2.500
ø 1.497
TYP
R.25
TYP
0.150
1.128
0.502
1.001
0.248
1.281
4.870
10.500
Right Side View
1.250
5.500
ø.75 Thru
4 Holes
ø1.75
Thru
R.125
4 Places

# CHAPTER EIGHT
# BLOCKS AND HATCH

## What is the Purpose of Blocks?

Throughout your daily workflow, you will frequently encounter shapes that require repetitive use. There are two approaches you can take: either manually draw the object each time you need it, or create a block by drawing it once and saving it as a single object. This block can then be inserted as many times as you want in the current file, as well as in other files.

**Using blocks in AutoCAD offers numerous advantages:**

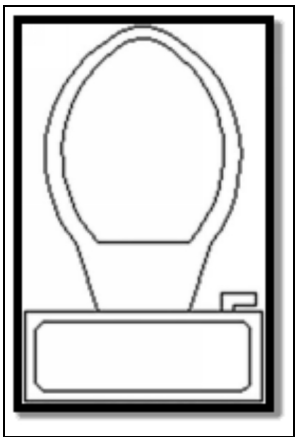
- Each block will be considered as a single object, resulting in a smaller file size.
- Implementing standardization within a company
- Efficiency in completing a drawing is crucial.
- Using the Design Center and Tool Palettes.

## Creating a block in AutoCAD

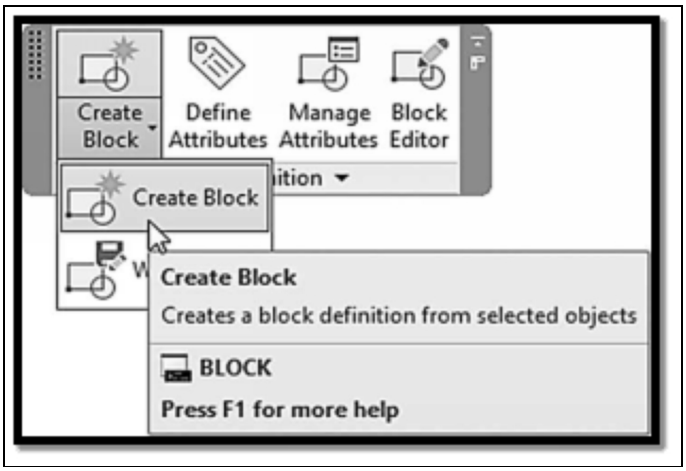
**To create a block, you need to follow these steps:**

- Begin by drafting the desired shape on layer 0 (zero). By setting the block to Layer 0, it will automatically adopt the properties (such as color, line type, and line weight) of the layer it is placed on.
- Ensure that you manage the "**Block unit**" setting in AutoCAD to allow for automatic scaling of blocks to the appropriate size in different drawings.
- Begin by sketching the desired shape with accurate measurements to create a block.

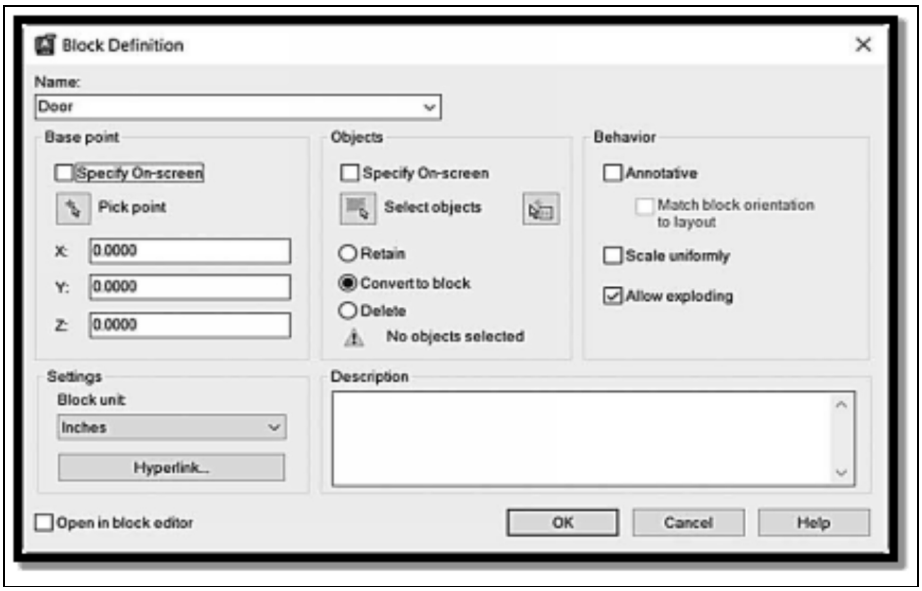
**Assuming the shape depicted below has been drawn:**



Once you're prepared, you can proceed with the following steps: navigate to the Insert tab, find the **Block Definition panel**, and click on the **Create Block button**.

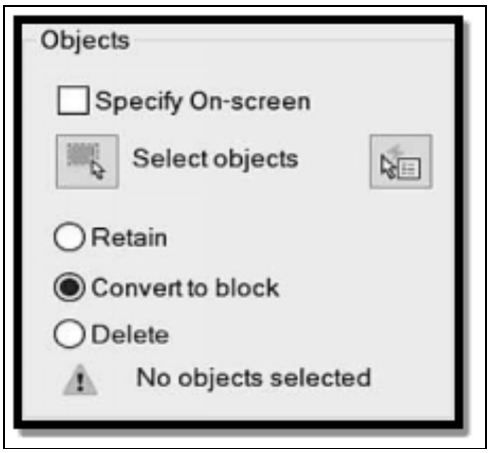



**A dialog box will appear on your screen.**

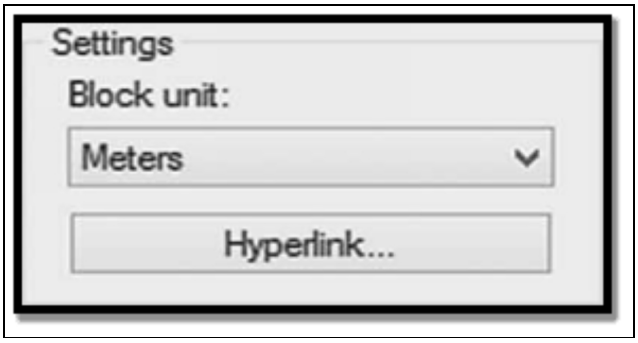



**Now, proceed with the following steps:**

- Enter the name of the block, ensuring it does not exceed 255 characters and only contains numbers, letters, hyphens, underscores, dollar signs, and spaces.
- When specifying the base point, you have two options. You can either enter the X, Y, and Z coordinates manually, or you can use the Pick Point button to select the base point graphically.
- Click on the "**Select objects**" button to choose the objects you want.
- After creating the necessary block from the drawn objects in AutoCAD, it is important to determine the next steps for handling these objects. When using AutoCAD, users have the option to choose from three different actions for objects: retaining them as they are, converting them to a block, or deleting them.
- Choose the desired options for the block, such as making it Annotative, enabling uniform scaling in both X and Y and allowing the block to be exploded at a later time.




- Choose the **Block unit**. Specify the unit conversion for each block to optimize AutoCAD's automatic scaling feature.



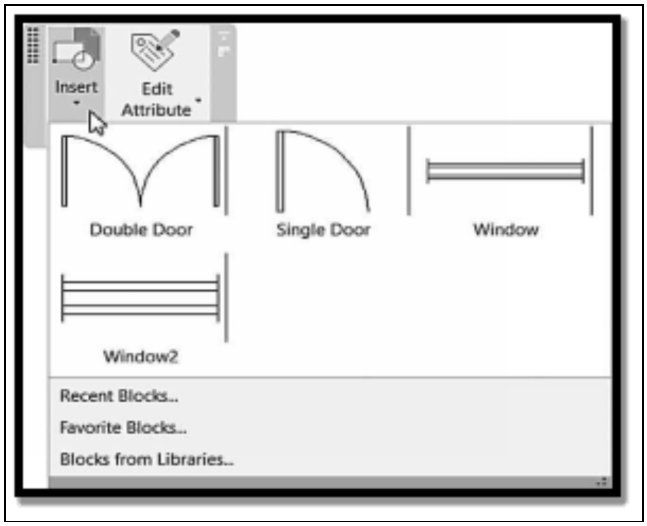

- Enter the block description.
- Disable the "**Open in a block editor**" checkbox as it is an advanced function primarily utilized for constructing dynamic blocks.
- After inputting all the necessary data, simply click on the OK button to proceed.

You can use the block by inserting a copy of its definition while keeping the original block definition unchanged.

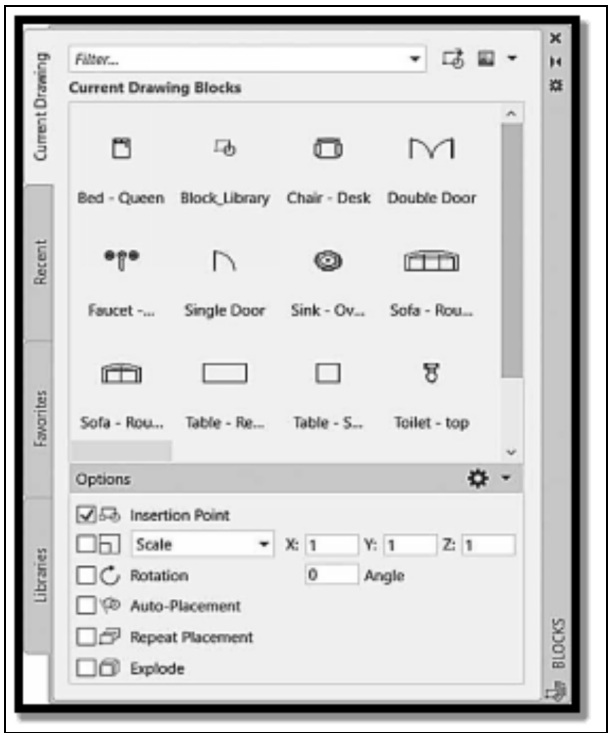
# How to Use (Insert) Blocks

Once the block has been created, the user can proceed to insert it into the current drawing. However, it is important to ensure that you are working on the correct layer and that the drawing is prepared to accommodate the block. For instance, make sure to create the door openings before inserting

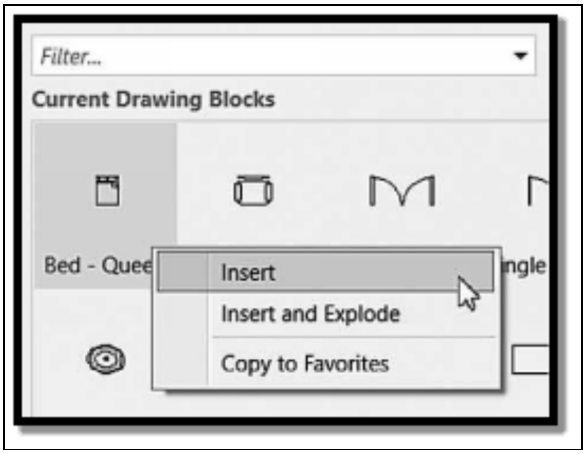
the door. Now you can proceed with the command by navigating to the Insert tab, finding the Block panel, and clicking on the **Insert button**:




After accessing the drawing, a comprehensive list of the existing blocks will be displayed for your convenience. With this approach, it is not possible to modify the scale or rotation angle. Inserting blocks in your drawing can be done quickly and efficiently. If you want to personalize the insertion process, you have the option to choose from Recent Blocks, Favorite Blocks, or Blocks from Libraries. **Once selected, a palette will be displayed:**

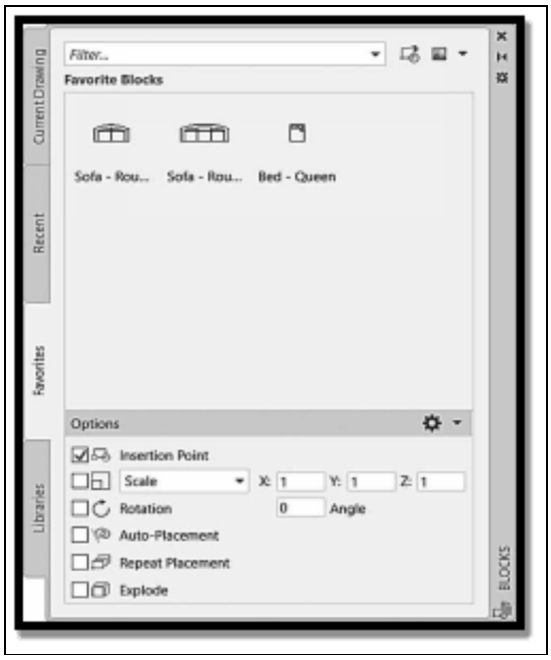



On the Current Drawing tab, you'll find a comprehensive list of the blocks present in this drawing. To ensure accurate placement of the block, it is important to provide the necessary details. These include the insertion point, scale, rotation angle, preference for auto-placement, preference for repeat placement, and the desired method of inserting the block (exploded or as one unit). You can either use the Specify On-Screen checkbox or manually input the required values. **When you perform a right-click on any block, a menu will appear:**



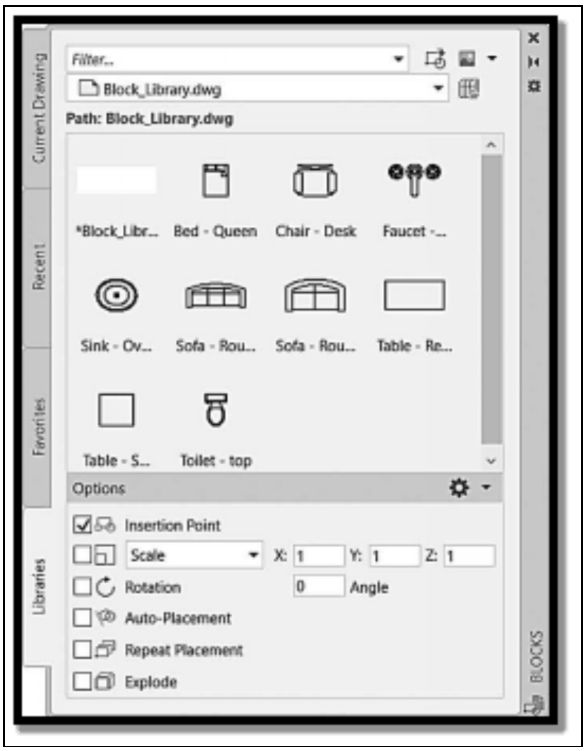
Within this menu, you have the option to insert the block, insert and explode it, or copy it to your favorites. By selecting the **Recent tab,** you can conveniently view all the recently inserted blocks, irrespective of the current drawing, as shown in the figure below. These elements are retained across different drawings and sessions. To remove a block from this tab, simply right-click on it and select "**Remove from Recent List**".

**Navigate to the Favorites tab to access the blocks that you have selected as favorites for convenient future use. Here, you will find the following:**



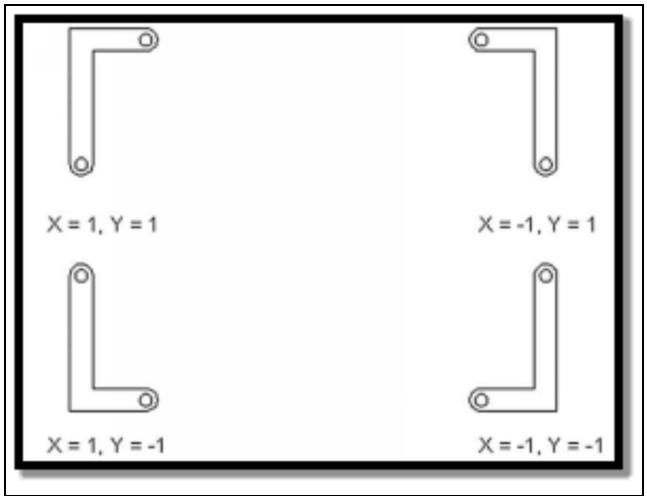

**After navigating to the Libraries tab, the user will be presented with the following:**

Select the book shape located in the top right corner. Then, indicate a file that contains the blocks you need or choose a folder that contains multiple files with the desired blocks. **These are the elements that will be visible:**




# Scale

When using the **Scale function**, it is possible to incorporate mirror images of the block by inputting negative values. Check the accompanying diagram:




When working with the Rotation angle in AutoCAD, it's important to keep in mind that counterclockwise (CCW) rotation is always considered positive.

# Auto-Placement

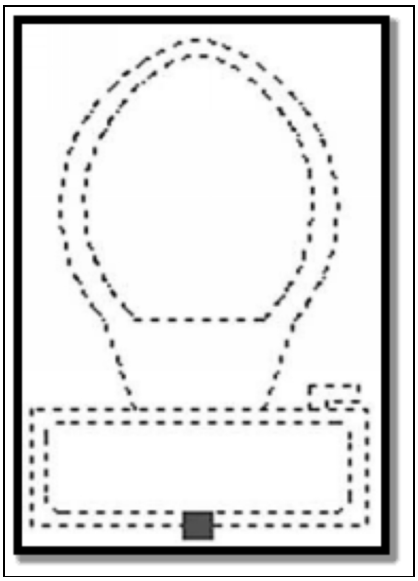
The Auto-Placement option in AutoCAD provides placement suggestions based on the previous placement of the block within the drawing. AutoCAD uses the knowledge of how the existing block instances are positioned in your drawing to intelligently propose the optimal placement for the same block.

# Repeat Placement

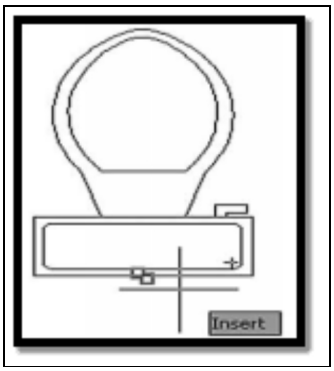
To insert a block multiple times in your drawing using the same command, simply enable the **Repeat Placement checkbox**. Right-click on the desired block and choose the **Insert option**. The block will then be inserted as many times as you need. This will significantly reduce the time required for the insertion process.

# Block Insertion Point OSNAP

**Once you have inserted a block incidence, simply click on it to view the following:**

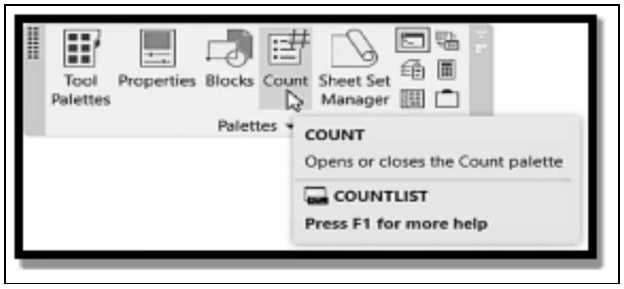


The entire block is considered a single unit, with only one grip, which is the insertion point that is highlighted. There is a particular OSNAP option available for snapping at this specific point, known as Insertion (or insert, depending on your perspective. Check the accompanying diagram:

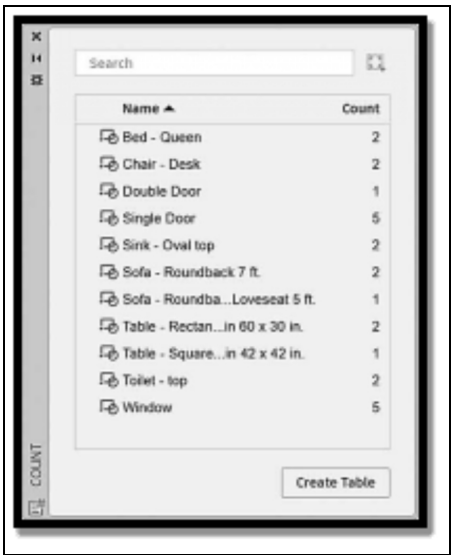



# Block Count

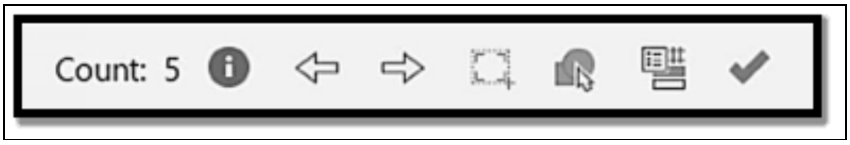
The command efficiently calclates the number of block instances in the current drawing with accuracy. You will be able to easily zoom in and navigate through the instances of any block in AutoCAD. Navigate to the **View tab** and find the **Palettes panel**. From there, simply click on the **Count button**.




**After opening the interface, you will be presented with the palette in question.**




To count the blocks within a drawing, you have two options. You can either use the Count command for the entire drawing, or you can specify a window by clicking the green upper right button to count the blocks within a specific area of the drawing. After selecting one of the blocks, you will notice a blue frame surrounding the screen, the block will be highlighted in green, and the Count toolbar will be displayed.




**Use this toolbar for the following tasks:**

- To zoom in for the block instances, utilize the left and right arrow keys.
- To limit your zooming, you have the option to specify a window within the drawing.
- Instances can be selected.
- Inserting a count field is a straightforward process.

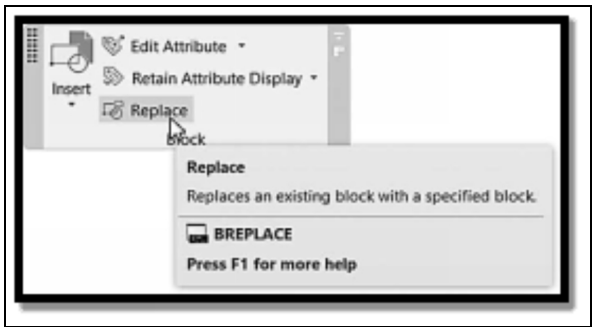
Click on the **Create Table button** located at the bottom of the Count tool palette. This action will display the following:

To create a table, simply select the blocks you want (one or more) and click on the **Insert button**. You will then receive the desired result.

| Item | Count |
|---|---|
| Single Door | 7 |
| Window | 5 |

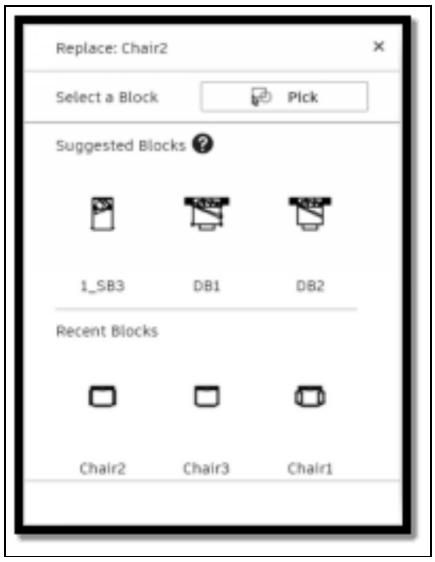
# Block Replace

This command allows you to effortlessly replace selected block references with a different block of your choice. It is possible to substitute one or multiple blocks in a drawing with a block of your choice, either from a list of recent or suggested blocks. Navigate to the **Insert tab**, find the Block panel, and select the **Replace button**:




**After opening the program, a prompt will be displayed:**

- Choose one or more blocks that you would like to replace:
- Choose the block you want to replace.
- Choose the block (or blocks) you want to replace. Note that you can only select blocks **with the same name. Once selected, a palette will appear with the following options:**



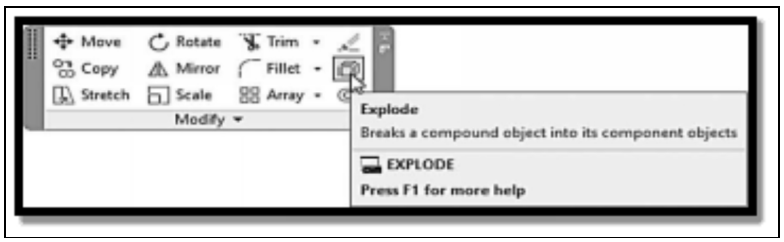

**There are three options available for you to choose from:**

- Choose a block that already exists in the current drawing.
- Choose one of the suggested blocks.
- Choose one of the most recent blocks.

# How to Explode Blocks and Convert Them to Files

# Exploding Blocks

Similar to the process of exploding polylines into lines and arcs, we can also apply the same technique to blocks. Using the explode command will restore the objects to their original state. It is strongly advised against this practice, as it is always recommended to keep blocks intact and avoid exploding them. **To execute the command, navigate to the Home tab, find the Modify panel, and click on the Explode button**:

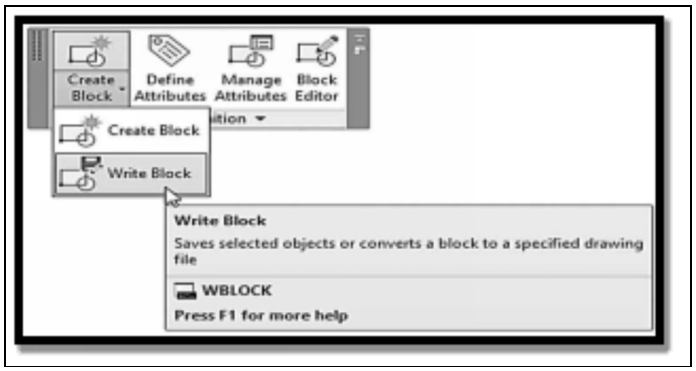



**A prompt will be displayed for you to see:**

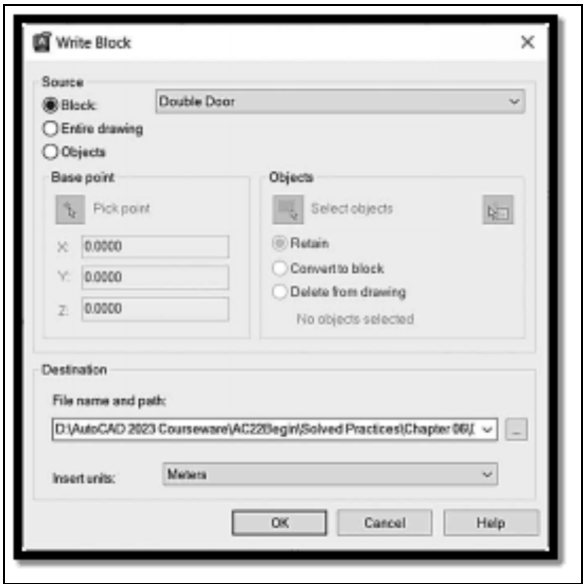
- Choose the objects you want to work with.
- Choose the blocks you want and hit [**Enter**] to complete the command.

# Converting Blocks to Files

In the past, it was common practice to convert blocks into separate files for use in other files. This practice became overshadowed by the introduction of the Design Center and Tool Palettes. To execute the command, navigate to the **Insert tab** and find **Block Definition**. From there, simply click on the **Write Block button**.




**A dialog box will be displayed to you.**




Choose the **Block option** from the Source menu. Indicate the name of the block. Specify the file name and path under the Destination input, and indicate the insert unit. Use the dialog box to effortlessly generate a file from either the entire drawing or specific objects within the current drawing.

# How to Hatch in AutoCAD 2025

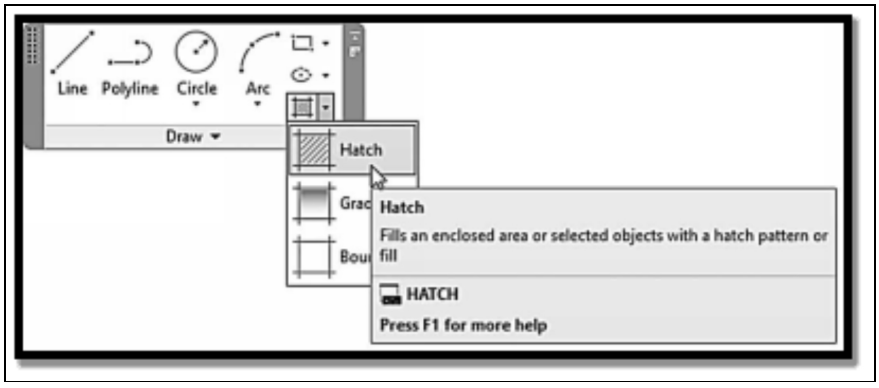
In AutoCAD, you can apply hatching to both closed and non-closed areas, allowing you to define the maximum distance as needed. AutoCAD includes two hatch pattern files: acad.pat and acadiso.pat. The extension for the hatch pattern files is *.PAT.

**There are four different types of patterns:**

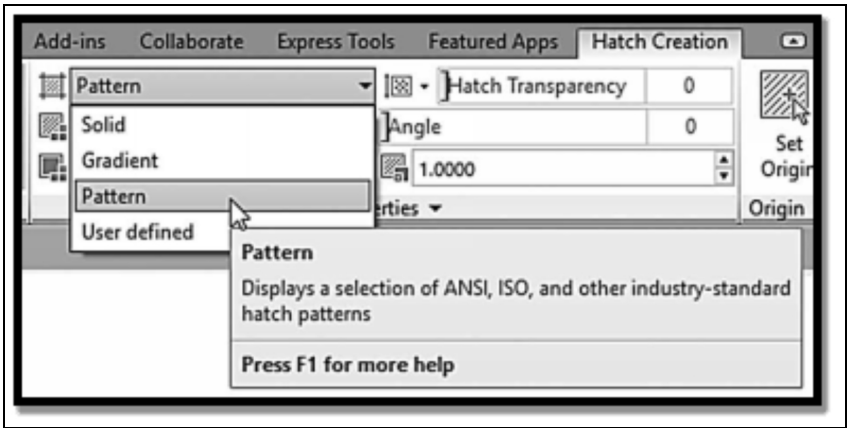
- A **solid pattern** is used to cover the area with a single color.
- **Gradients** in AutoCAD allow for the blending of two colors in various ways.
- **Pattern** (multiple pre-set patterns)
- **User-defined patterns** can be created easily, such as parallel lines.

# Using the Hatch Command

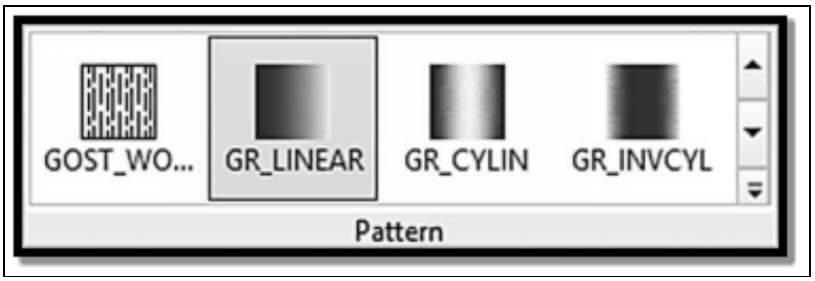
This command is used to apply hatches to the drawing and allows for complete control over their properties. The preview is generated instantly. To access the Hatch command, navigate to the **Home tab** and find the **Draw panel**. From there, simply click on the **Hatch button**.




A new context tab called Hatch Creation will be added to the ribbons. There are multiple panels that we will discuss individually as we go along. Begin by finding the Properties panel. In the top left corner, choose the **Hatch Type** as demonstrated:



After selecting the Hatch Type, you can find the Pattern panel. AutoCAD will automatically navigate you to the first pattern in the selected type. When you choose Gradient as the Hatch Type, the initial pattern in the Pattern panel will be **GR_LINEAR**, which happens to be the first pattern in the gradient patterns.




Now, navigate effortlessly to the desired area to be hatched, and witness the seamless filling of the area.

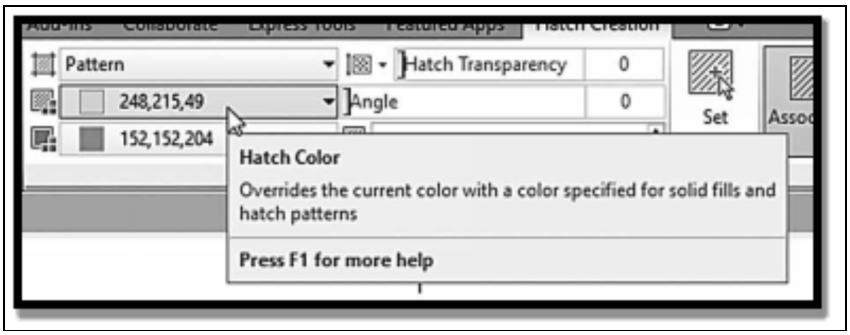
**Currently, there are two options available:**

- If you are satisfied with the outcome, select the desired area and navigate to the Close panel. From there, simply click on the option labeled **Close Hatch Creation**.
- If you're not happy with the result, you'll need to manipulate the properties of the hatch to try and fix it.

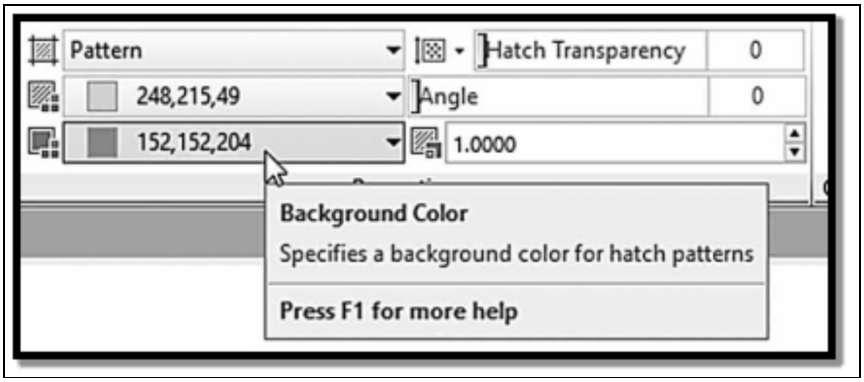
# How to Control Hatch Properties

If you have selected an area and are unsatisfied with the outcome, you can modify the properties of the hatch to achieve the desired result. All of these functions can be found in the Properties panel.
**Here are the details:**

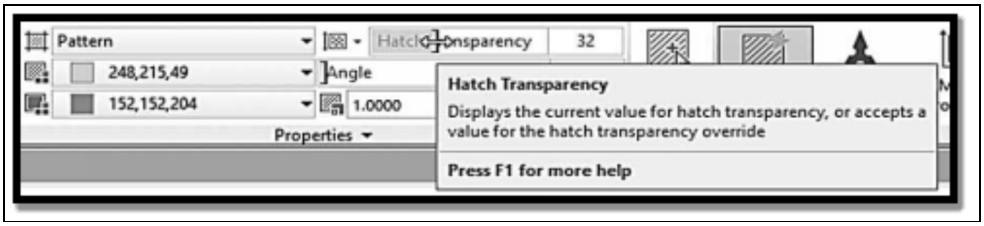
- **Hatch Color**: Specify the color of the hatch or leave it as "**Use Current**" for the hatch color.



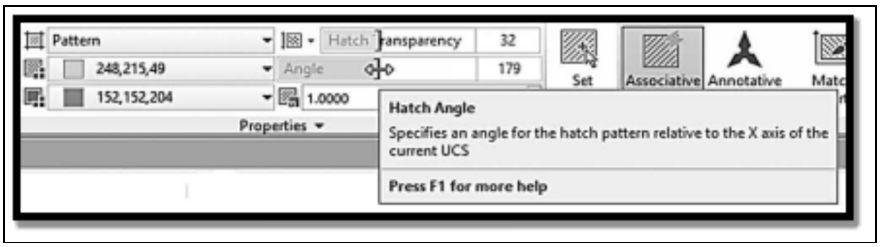

- **Background Color:** Specify the desired background color or choose "**None**" if no color is preferred.



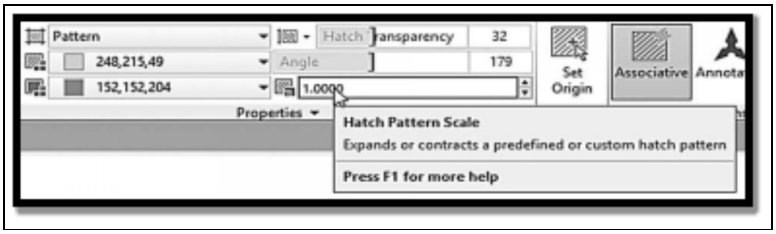

- **Transparency**: By default, all colors in AutoCAD are displayed with their normal intensity. However, you have the option to adjust the transparency value (up to a maximum of 90) to decrease the intensity of the color. This applies to both hatch colors and background colors.



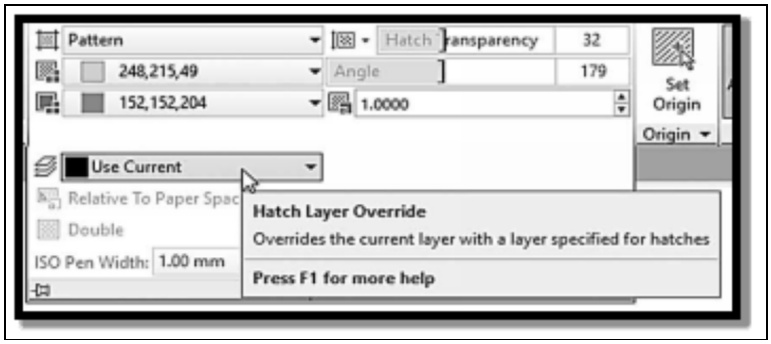

- **Angle**: Specify the angle of the hatch pattern. Note that this feature does not have any effect with Solid hatching.




- **Scale**: Enter the desired scale or spacing for the selected hatch pattern.




- **Hatch Layer Override**: With the Hatch Layer Override feature, you can specify a layer for the hatch to reside in, regardless of the current layer. By default, the hatch will be placed in the current layer, but this function allows for more control over the hatch's placement.

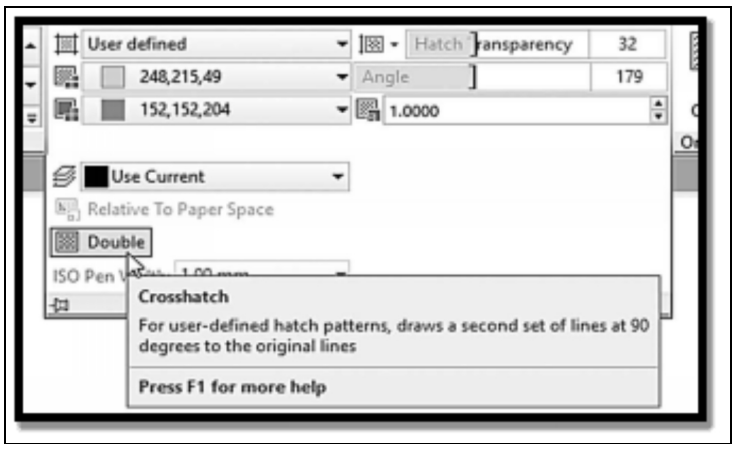



# Important

**One way to create a new layer and set it as the default layer for the hatch is by utilizing the system variable HPLAYER.**

- **Double**: When working with a User-defined hatch type, you have the option to double the hatch. Keep in mind that this option is only applicable in this specific scenario. It determines

the orientation of the lines, either in a single direction or in a crosshatch pattern.



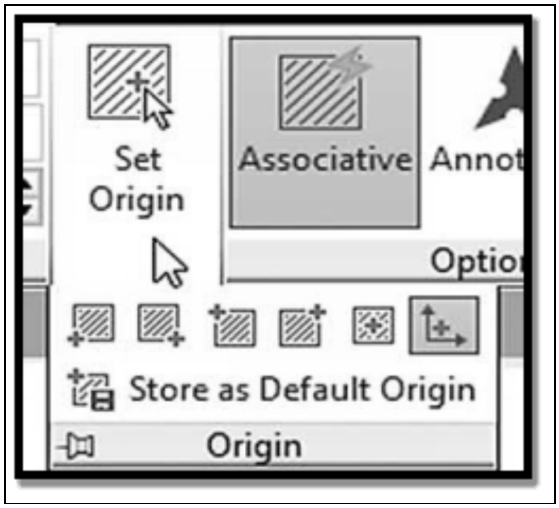

# How to Control Hatch Options

These options have a significant impact on the hatching process. By using these options, you can effectively hatch open areas with precision (Gap Tolerance) or generate distinct hatches for various purposes. **These are just a few examples of the capabilities at your disposal:**

# How to Specify Hatch Origin

If you want to customize the starting point of a hatch pattern in AutoCAD, rather than using the default settings, you can manually set the Hatch Origin. By default, AutoCAD utilizes the coordinate (0,0) as the origin for hatches. Consequently, there is uncertainty regarding the accurate display of

your hatch. To control the Hatch Origin, ensure that you are in the Hatch Creation context tab. Look for the Origin panel, where you will find the following options:

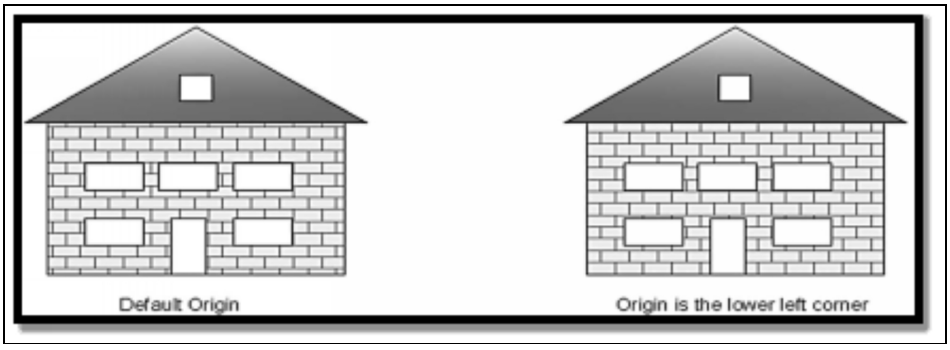



The button you're looking for is called "**Set Origin**." When you click on it, you will be prompted to:

# Specify the origin point

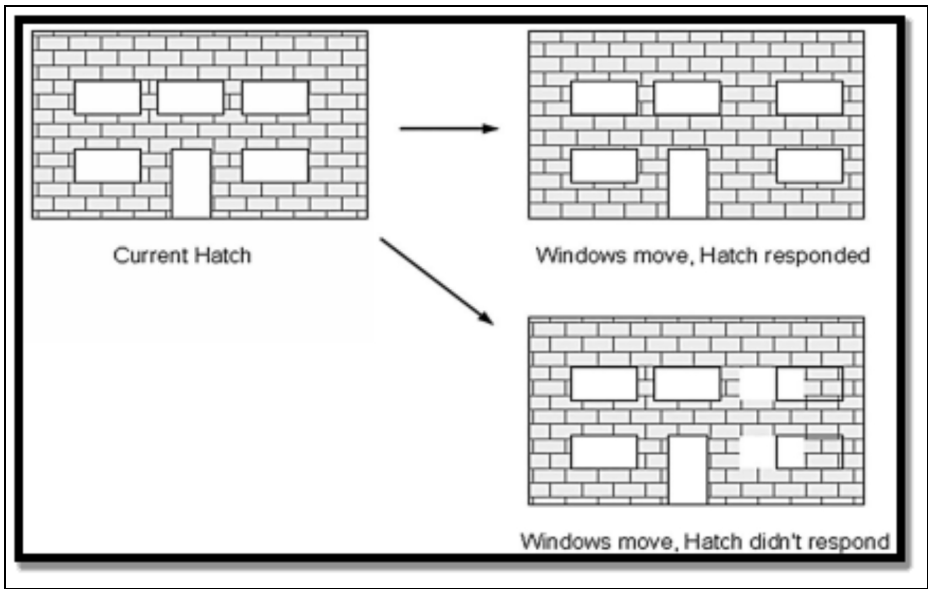
Indicate the desired point or use the preexisting predefined points such as the lower left corner or upper left corner. It is possible to save the selected point for future reference, rather than relying on the default coordinates of 0,0.

**Take a look at the following example:**



# Creating Associative Hatching

In AutoCAD, the hatch feature is designed to be associative, allowing it to intelligently recognize and fill the boundaries it is applied to. When the boundary is modified, the hatch will accurately adjust accordingly.
**Take a look at the illustration provided:**

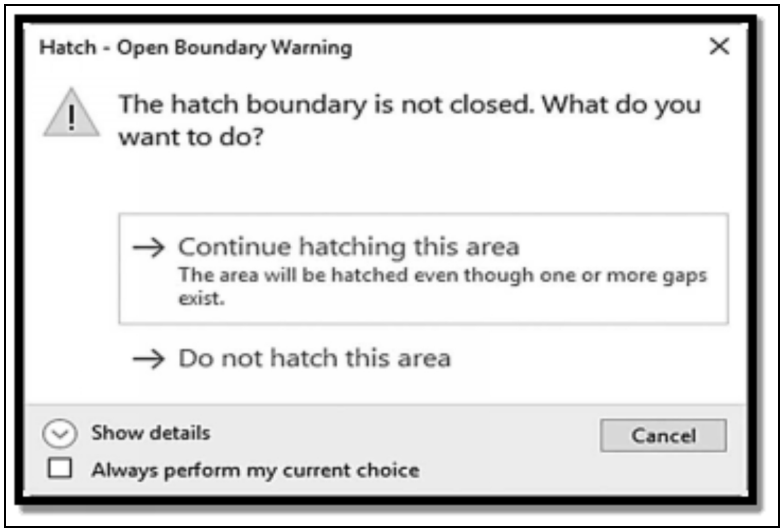



# How to Hatch an Open Area

By default, AutoCAD will hatch only closed areas. Fortunately, AutoCAD allows you to easily hatch an area with an opening. To enable hatching in an open area in AutoCAD, you can easily configure the **Gap Tolerance**. This parameter determines the maximum allowable opening size and any area with an opening larger than this value will not be hatched.

When hatching this area, the preview will not be displayed. To proceed, simply click inside the opened area.

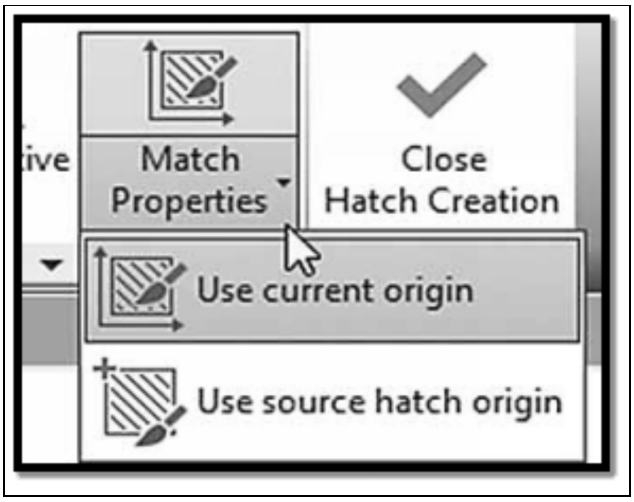
**A warning message will then appear:**




You have the option to either proceed with hatching the open area or skip this step.

# How to Use Match Properties to Create Similar Hatches

This feature allows you to effortlessly generate a duplicate hatch that perfectly matches an existing one. The duplicate hatch will be placed in the same layer and will retain all the original properties, such as angle, scale, transparency, and more. This feature offers two buttons: "**Use current origin**" and "**Use source hatch origin**".



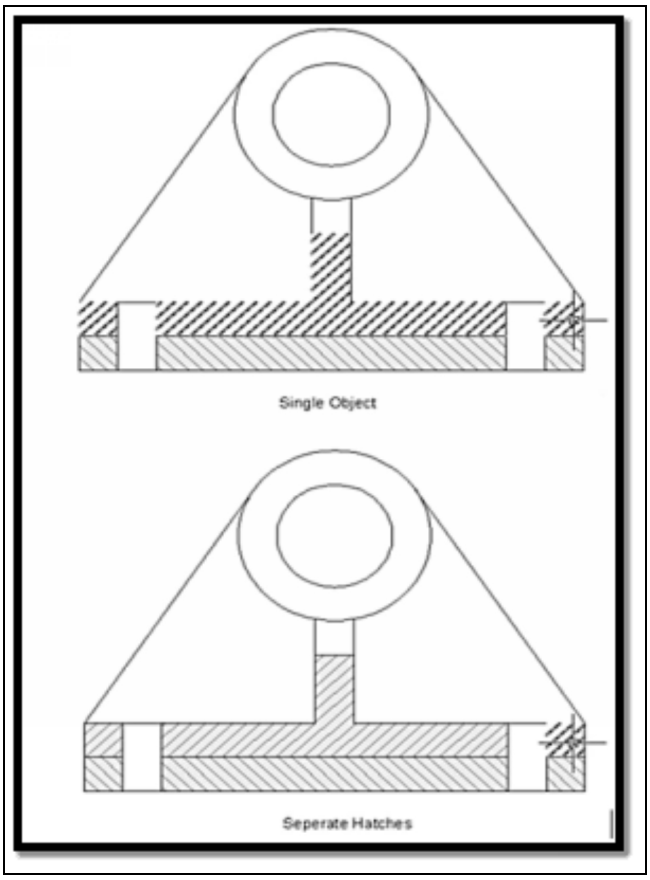

**Click on the Match Properties button to access the prompt that will appear:**
Select the hatch object:
Choose the hatch object you want to replicate, and a prompt will appear:

Pick internal point or [Select objects/seTtings]:

Click within the designated region. Continuously choose the desired region, and once finished, hit [**Enter**] to end the command.

# How to Create Separate Hatches in the Same Command

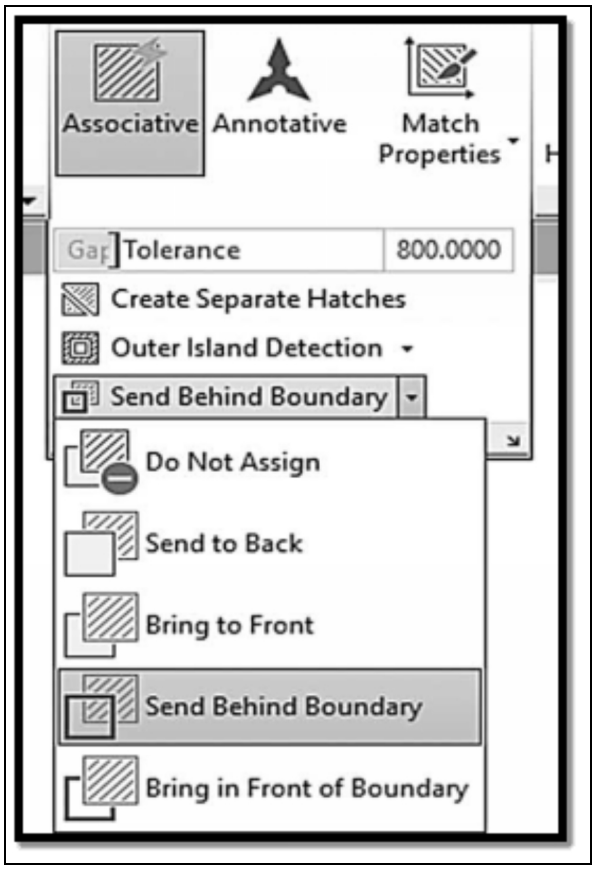
When using the same command, if you have hatched multiple distinct areas, AutoCAD will treat them as a unified hatch, or a single object. To customize the default setting, you can indicate to AutoCAD that you require distinct hatches for different areas. Just activate this button.




# How to Set Hatch Draw Order

When hatching in AutoCAD, you can adjust the draw order of the hatch to other objects.

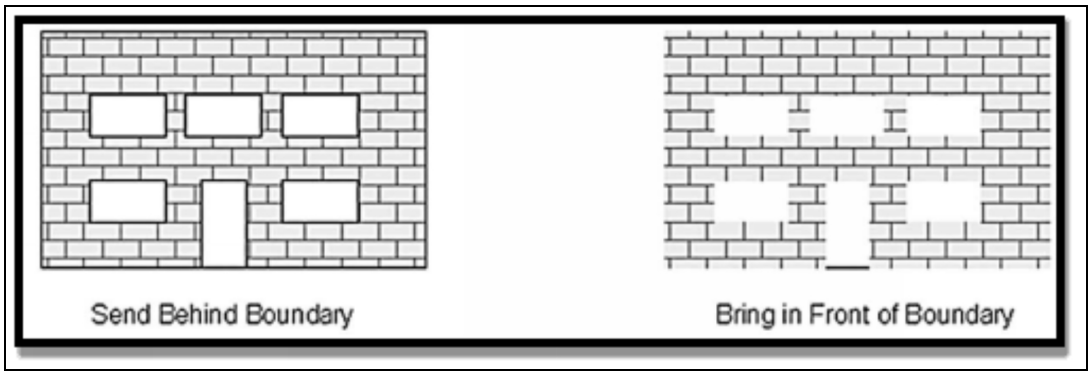
**There are five options available for you to choose from:**




**Here are the five cases:**

- Send to Back
- Do Not Assign, use the default
- Bring in Front of the Boundary
- Send Behind Boundary

**Take a look at the following example:**



# CHAPTER NINE
# TEXT

In this chapter, you will gain a comprehensive understanding of navigating through AutoCAD's TEXT and MTEXT commands. Furthermore, you will come across numerous editing commands that are commonly used with text but are equally crucial for editing other objects.

## How to Enter Single-Line Text with Justification Options

Two commands in AutoCAD allow you to input text into a drawing. When using **TEXT** in AutoCAD, you can effortlessly input single lines of text and see them appear on the screen as you type. **MTEXT** enables the user to input multiple lines of text using a dedicated text editor and then arrange them within a designated area of the drawing. There is a wide range of options available for placing and formatting text in both commands. Creating and applying named styles to text is a great way to ensure a consistent appearance throughout your drawing. If you're already well-versed in inputting single lines of text that are aligned to the left, this should be a breeze for you. Now, you can start incorporating different types of single-line text into your design.

- For this exercise, start by creating a new drawing with B-size (18 × 12) limits. Make sure to turn off the grid display. Then, draw a 4.00-unit horizontal line starting at the coordinates (1,1). Next, generate five duplicates of the line, each separated by a distance of 2.00 units.

**These lines serve as a reference for this exercise and are not necessary for drawing text. Begin by conducting a brief overview of the left-justified text.**

- To access the Single Line text tool, navigate to the **Annotate tab**, then go to the **Text** panel and click on the **Text** drop-down menu.
- Indicate the starting point on the left side of the upper line.
- Simply press <**Enter**> to accept the default height of **0.2000**.
- Press the **Enter key** to accept the default angle of 0.
- Press the **Left key** on your keyboard and then hit Enter. It is important to observe that the text cursor will move to the next line when the <**Enter**> key is pressed.
- Set the text alignment to justified. Once again, the text cursor unexpectedly moves downward, and yet another prompt to enter text pops up. To exit the command, simply press the <**Enter**> key when prompted.
- To exit the command, simply press the **Enter** key. After executing this step, you will be directed back to the command-line prompt.

**Here's a helpful suggestion:** Text in AutoCAD is often organized by placing it on designated layers. Use the text layer override setting to effortlessly place new annotations on a designated text layer, eliminating the need to manually switch layers beforehand. To modify the text layer override setting, you can follow these steps: To access the desired options, navigate to the Annotate tab and expand the **Text panel**. From there, choose a layer from the **Text Layer Override** drop-down list. To specify the layer you want to use, simply enter the **TEXTLAYER system** variable at the command line and type in the name of the desired layer. Before exploring alternative text alignment options, it is important to familiarize yourself with an additional attribute of **TEXT**.

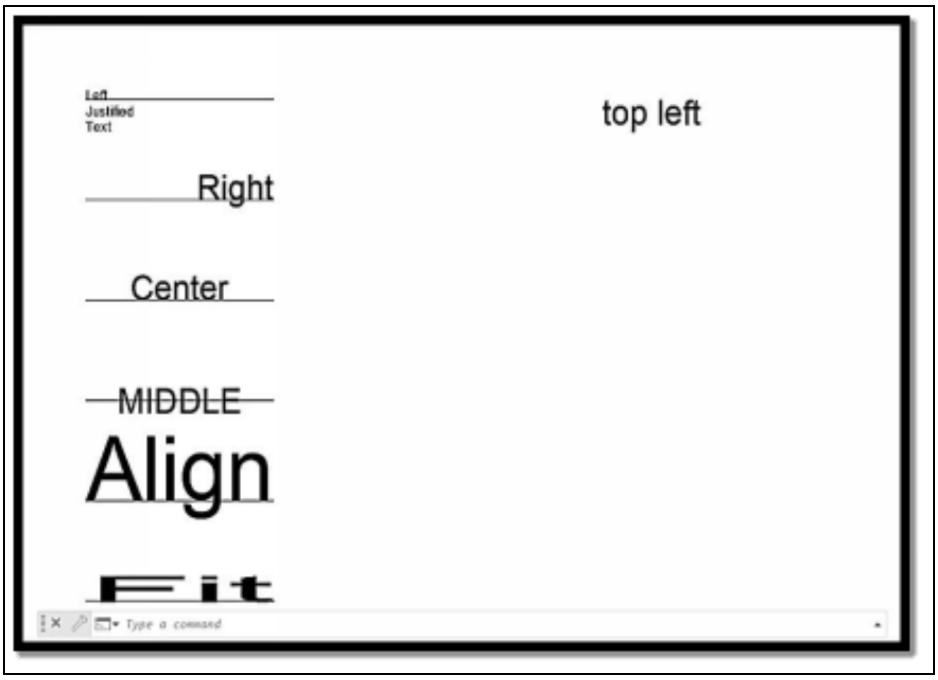
- Press the **Enter key** to repeat the **TEXT command**. Pressing <**Enter**> again at this point, without specifying a new start point or selecting a justification option, will take you back to the Enter text prompt as if you had never left the command. Give it a shot.
- Press the <**Enter**> key once more. After encountering the Enter text prompt in the command line, the text cursor will reappear just below the word Justified.
- Enter **Text** <**Enter**>. When you encounter the "**Specify start point**" prompt in the command line, it is possible to relocate the text cursor to a different point. Give it a shot.

- Indicate a new starting point at any location within the drawing. The text cursor will automatically move to the new point that you specify.
- To exit the command, simply press the **Enter key**.

**Here's a helpful suggestion:**

- At times, it may be necessary to align or evenly space multiple text objects. You can achieve this using the **TEXTALIGN command**.
- To align text in AutoCAD, go to the **Annotate tab** and select **Text**. From there, choose **Text Align**.
- Choose the text objects that you want to align.
- Choose the text object that you wish to align with all the other selected text objects.
- Provide the coordinates for the second alignment point.

Next, we will explore the various text justification options, starting with right-justified text. When working with the text in this exercise, it is important to also consider the change in height.



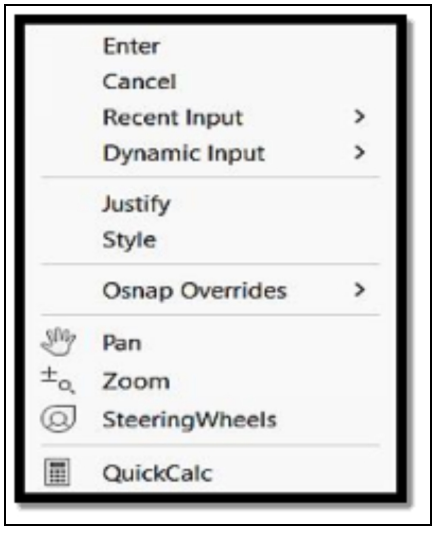

**Here are some examples of text justification options:**

| TEXT JUSTIFICATION | |
|---|---|
| START POINT TYPE ABBREVIATION | TEXT POSITION +INDICATES START POINT OR PICK POINT |
| A | ALIGN |
| F | FIT |
| C | CENTER |
| M | MIDDLE |
| R | RIGHT |
| TL | TOP LEFT |
| TC | TOP CENTER |
| TR | TOP RIGHT |
| ML | MIDDLE LEFT |
| MC | MIDDLE CENTER |
| MR | MIDDLE RIGHT |
| BL | BOTTOM LEFT |
| BC | BOTTOM CENTER |
| BR | BOTTOM RIGHT |

# Right-Justified Text

**When creating right-justified text in AutoCAD, you start from an endpoint and move backward, aligning the text from right to left.**

- Execute the **TEXT command** again.
- To access the Justify command, simply right-click and choose it from the shortcut menu.




**When Dynamic Input is active, it will display a dynamic input list.**

- To select "**Right**" from the menu, make sure that Dynamic Input is open. Alternatively, choose "**Right**" from the command line.

Instead of a start point, the software prompts you to specify an endpoint:
Indicate the endpoint of the text baseline on the right side:

- Indicate the endpoint of the second line. Provide the desired height for the text. Now, adjust the height to 0.50.
- Enter **.5** and press the **Enter key**.
- Press the **Enter key** to maintain a rotation angle of 0 degrees. Observe the enlarged text cursor positioned at the right side of the second line.
- Press the **Enter key** to execute the command. To align the word "**Right**" to the right on the second line, you can use the right-justification feature.
- To exit the command, simply press the **Enter key**.

Ensure that the second line of text in your drawing is positioned in a right-justified manner.

# Centered Text

**Text that is centered is justified from the bottom center of the text.**

- Execute the **TEXT command** again.
- Choose the **Justify option** from the command line.
- Choose the "**Center**" option either from the command line or the Dynamic Input list.

AutoCAD provides prompts:
Specify the center point for the text:

- Specify the midpoint of the third line.
- To keep the current height, simply press <**Enter**>. The current height is currently set to 0.50.
- Press the **Enter key** to maintain a rotation angle of 0 degrees.
- Press the **Enter key** after typing "**Center**".
- Press the **Enter key** once more to finalize the command.

The term "**Center**" should now be aligned in the center.

# Middle Text

The alignment of the middle text is centered both horizontally and vertically, rather than being centered from the bottom. In the past, you would have chosen justification options by using the command line or shortcut menu. However, now you can save time by simply typing in the initial for the desired option. Additionally, it is customary to input this text using uppercase letters, as is typical in the majority of drawings.

- Repeat the **TEXT**.
- Enter "**m**" and press the **Enter key**.

AutoCAD provides prompts:
Specify the midpoint of the text:

- Specify the midpoint of the fourth line.
- Press the **Enter key** to keep the current height at 0.50.
- Press the **Enter key** to maintain a rotation of 0 degrees.
- To enter text in all capital letters, simply press the **Caps Lock key** on your keyboard.
- Press the **Enter key** after typing "**MIDDLE**".
- Press the **Caps Lock key** on your keyboard to deactivate the capital letters.
- Press the <**Enter**> key once more to finalize the command.

Take note of the distinction between center and middle. The center point is located at the midpoint of the baseline below the text. The middle point is located within the text, causing the line to pass through it.

# Aligned Text

Text can be aligned by specifying two points. The height of the text is determined based on the relative distance between the two points, and the text is then rendered along the line connecting the two points.

- Repeat the **TEXT**.
- Press the **Enter key**.

AutoCAD prompts:
Specify the initial point of the text baseline:

- Specify the starting point of the fifth line.

AutoCAD is requesting input for an additional point.
Specify the second endpoint of the text baseline.

- Indicate the endpoint of the fifth line.

It is important to note that the prompt for height is missing. AutoCAD uses a precise calculation to determine the height based on the distance between the selected points. Additionally, there is no need for a prompt regarding the angle as the program automatically calculates the angle between the two points, in this particular scenario, 0.

- Press the **Enter key** after typing "**Align**". At first glance, the text may seem oversized. However, as you start typing, the text size dynamically adjusts with each letter you add.
- Press the **Enter key** once more to finalize the command. Take note that the text is automatically adjusted to fit the area between the two points you have chosen.

# Text Drawn to Fit Between Two Points

**The Fit option functions similarly to the Align option, with the distinction that it preserves the designated text height.**

- Repeat **TEXT**.
- Press the letter "**f**" followed by the **Enter key**. When using this feature, you will be prompted to specify two points, similar to the Align option.
- Indicate the starting point of the sixth line.
- Indicate the endpoint of the sixth line.
- Press the **Enter key** to keep the current height. Similar to the Align option, there is no prompt to specify the angle of rotation.
- Press the **Enter key** after typing "**Fit**". As you type, the text size will be adjusted accordingly, with the width being the only

aspect that changes.

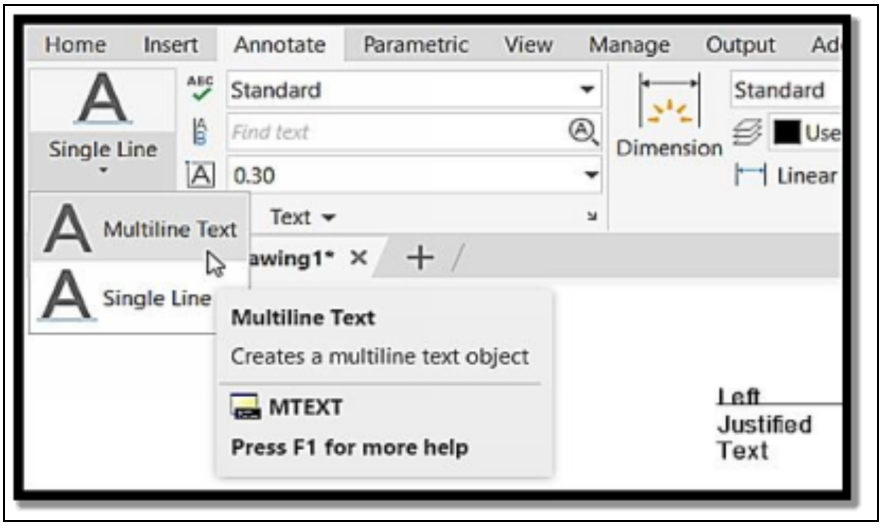
- Press the **Enter key** once more to finalize the command.

In this case, the text is expanded horizontally to occupy the entire line while maintaining its original height. Understanding the distinction between Fit and Align is crucial. Within the Align option, the text height is determined by the width that is displayed. When using Fit, the height you specify will remain the same, and the text will be adjusted to fit the available space by stretching or compressing it.

# How to Enter Multiline Text Using MTEXT

With the MTEXT command, you can effortlessly generate multiple lines of text and consolidate them into a single object using a text editor. Furthermore, you can easily position this object within a designated window in your drawing. Similar to **TEXT**, **MTEXT** offers nine different options for text justification and its own unique set of character codes.
**To start, you can create a basic block of text aligned to the left.**

- To access the Multiline Text tool, navigate to the Text panel of the Annotate tab and select it from the Text drop-down list.




**A prompt is displayed in the command line interface:**
Specify first corner:
Additionally, it is important to observe that a multiline text symbol, specifically "**abc**", has been included within the crosshairs. Essentially, **MTEXT** allows you to specify the width of a set of text lines that you

generate in a dedicated text editor. When you enter text in AutoCAD, the software automatically formats it to fit the specified width and justification options, and then it is drawn on the screen. There are multiple ways to define the width. By default, a common approach to indicate width in AutoCAD is to create a window in the drawing. However, it is important to note that this method can sometimes be misleading. The **MTEXT** command does not try to fit the entire text within the window, but rather only within its width. The initial location of the window serves as the insertion point for the text. The utilization of the insertion point in AutoCAD is contingent upon the chosen justification option. When specifying the second window point, it is important to consider the width and the desired text flow direction. This will determine whether the text lines should be drawn above or below, and to the left or right of the insertion point. The interpretation of this is also influenced by the chosen justification option.

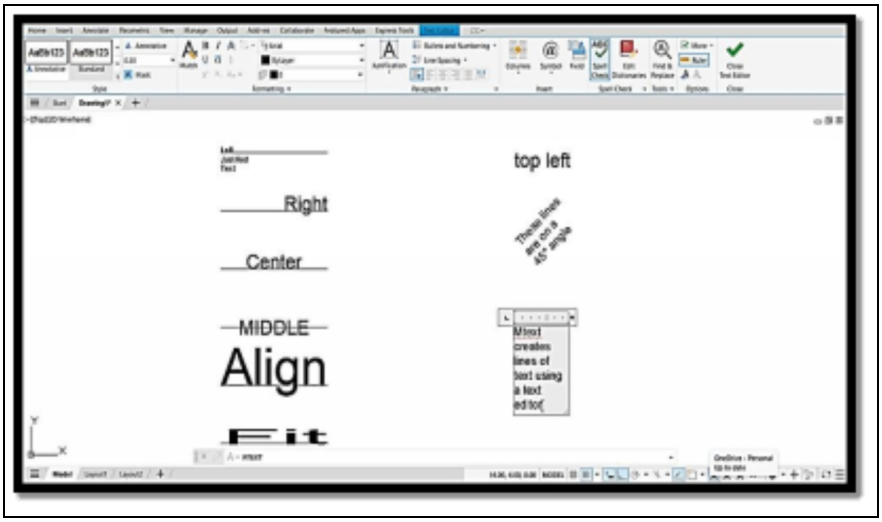
- Choose a suitable location for the insertion point within the drawing area, preferably near the coordinates (**12.00, 5.00**).

When initiating a command in AutoCAD, a window is created at the specified point, followed by a new prompt. Specify opposite corner or [Height/Justify/Line spacing/Rotation/Style/Width/Columns]:
Proceed with the default settings by indicating the coordinates of the second corner. For this exercise, a window with a width of 2.00 units is drawn downwards and to the right from the insertion point.

- Indicate the coordinates for the second point, which is located 2.00 units to the right and approximately 1.00 units below the insertion point.

When specifying the opposite corner, the Multiline Text In-place Editor in AutoCAD will automatically open at the insertion point in your drawing. Simultaneously, the Text Editor contextual tab is incorporated into the ribbon. The panels on the Text Editor tab provide you with the ability to modify text styles and fonts, as well as text height and various other text attributes like bolding and underlining. When entering text in the editor, it will wrap around based on the specified width, aligning with how it will be displayed in the drawing. The width is indicated by the small ruler located

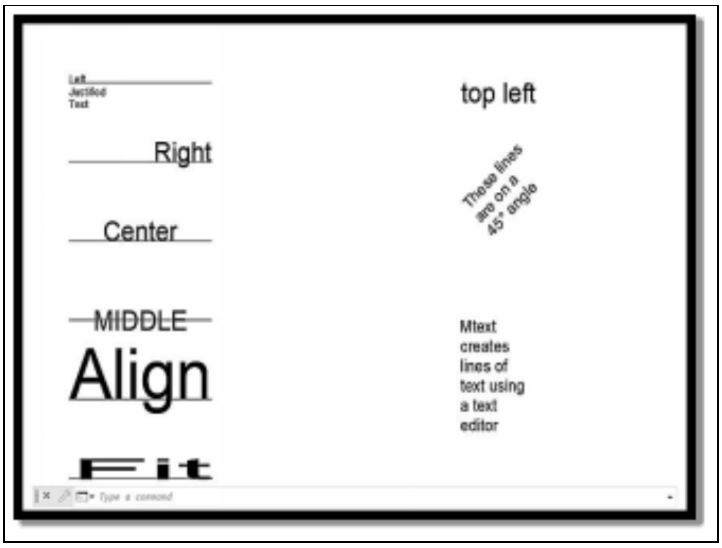
at the top of the text editor. By utilizing the small L located along the left side of the ruler, you can establish first-line indent and hanging indent tabs to effectively format paragraphs. When the cursor is placed on the right side of the ruler, a double-arrow cursor appears, allowing you to easily adjust the window width. When the cursor is positioned at the bottom edge of the text editor, a pair of vertical arrows will appear, allowing you to easily modify the window's height.




The text editor is positioned precisely at the location of the text, displaying the text in the editor at the exact size it will appear in the drawing. However, if this would result in the text being either too large or too small for easy editing, adjustments are made accordingly. When encountering such situations, the text size is automatically adjusted to a suitable size for editing purposes. The true size is then revealed once the command is executed.

- The Mtext command in AutoCAD allows you to create lines of text using a text editor. The text is displayed in the text editor.
- Choose a point outside the text editor to exit the editor and finalize the command.

You are back in the drawing window. The Annotate tab has been set as the current tab once more, and the new text has been added. Take note that the text you entered has been adjusted to fit within the 2.00-width window. The height of the text remains at 0.30, while the defined height of the text window at 1.00 has been disregarded.



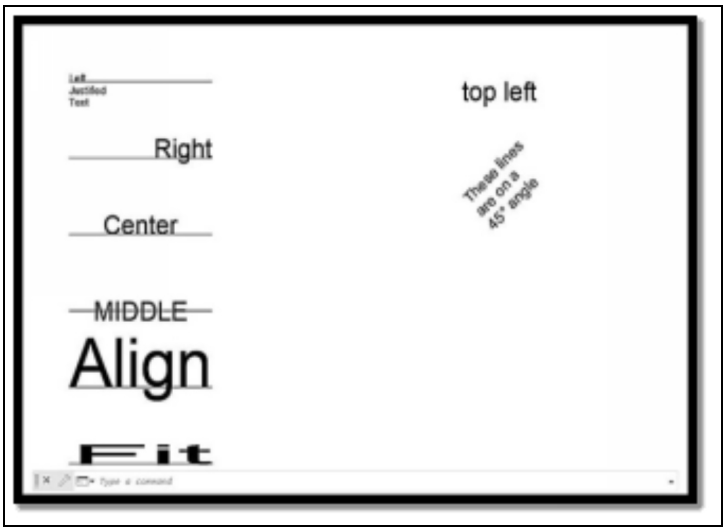

Now, let's delve into additional options for justifying multiline text (**mtext**). First, you can perform some basic text editing using **TEXTEDIT**.

**Here's a helpful suggestion:**

You can combine multiple single-line and multiline text (**mtext**) objects into a single **mtext** object. To merge multiple text objects, enter the command "**txt2mtxt**" at the prompt and follow the instructions.

# How to Enter Text on an Angle and Text Using Character Codes

In this section, we will delve deeper into TEXT by demonstrating how to enter multiple lines at an angle and incorporate special character symbols into your designs. To achieve the desired effect, you can create three lines of left-justified text, positioning them one below the other and rotating them at a 45° angle.




- Execute the **TEXT command** again. AutoCAD remembers your most recent selection of options, so if you haven't used any other text options, you will be prompted for a top-left point. This option is suitable for this exercise, but it is not essential.

**Specify the top-left point of text or (Justify/Style):**

- Specify a point near the coordinates (**12.00,8.00**).
- To specify a smaller text size, simply enter .3 and press **Enter**.
- Press the number 45 followed by the **Enter key** to establish an angle of 45 degrees. Observe that the text cursor is displayed at the designated angle.

- Enter **These lines <Enter>.** The text is displayed in your drawing at a 45° angle, and the cursor smoothly transitions to the next line. Take note that the text cursor remains at the designated angle.
- Specify the type of object by pressing the **Enter key**.

# The Degree Symbol and Other Special Characters

The following line includes the symbol for degrees. Adding a character that is not available on your keyboard to a single-line text object in AutoCAD can be done using a special method provided by the software. Enter the text using the %% symbols, as demonstrated in the initial step.

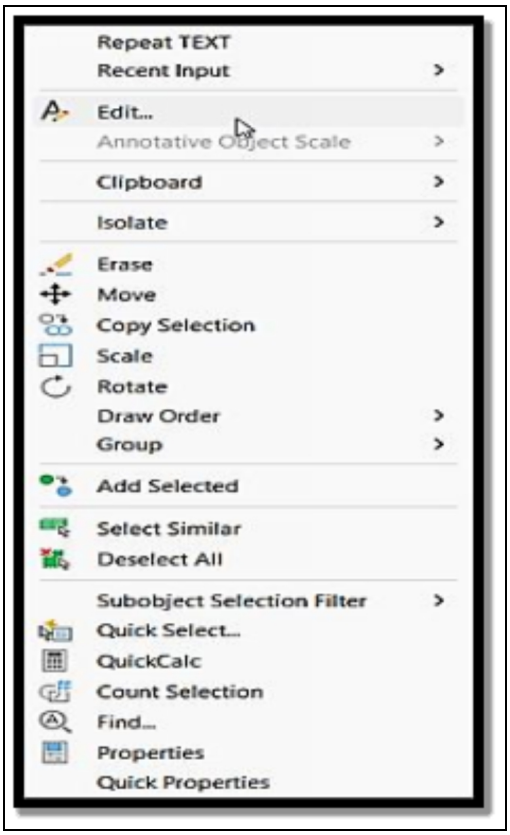
| Control Codes and Special Characters | |
|---|---|
| Type at Text Prompt | Text in Drawing |
| %%OOVERSCORE | OVERSCORE |
| %%UUNDERSCORE | UNDERSCORE |
| 180%%D | 180° |
| 2.00 %%P.01 | 2.00 ±.01 |
| %%C4.00 | Ø4.00 |
| 37.5%%% | 37.5% |

- Enter the value as 45%%d. When you type percent symbols in the text editor, they will be displayed directly as you have entered them. When you enter the letter "**d**" in AutoCAD, it is automatically converted and displayed as a degree symbol.
- To insert an angle in AutoCAD, simply add a space and then type the angle value, followed by pressing the **Enter key.**
- Press the **Enter key** once more to finalize the command sequence.

# How to Edit Text in Place with TEXTEDIT

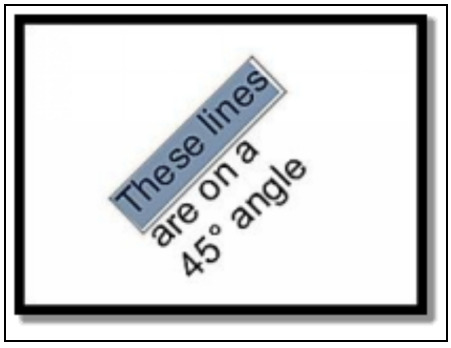
There are multiple techniques available for modifying the existing text in your drawing. Modifying text in AutoCAD involves adjusting elements like layer, style, and justification, as well as making changes to wording and spelling. To make simple changes to text wording, you can use the **TEXTEDIT** command. This command can be accessed by selecting a text object and choosing **Edit/Mtext Edit** from the shortcut menu or by double-clicking on a text object. For making property changes, you can use the Quick Properties palette, which will be discussed in the following section. In this exercise, you will learn how to perform basic text editing using **TEXTEDIT**. Begin by choosing the initial line of the angled text.

- To select the desired text, simply click on any of the letters within the phrase "**These lines".** The highlighted text and the appearance of two grips at the start point indicate the selection of this single line.
- To access the shortcut menu, simply perform a right-click.



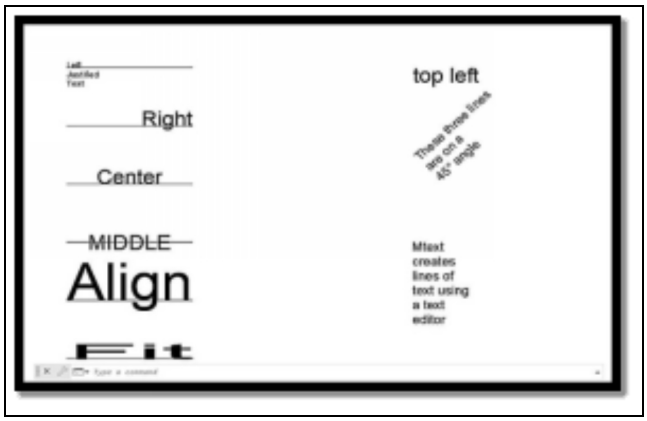

- Choose the **Edit option** from the shortcut menu, as demonstrated.

After making the selection, the shortcut menu will vanish and the highlighted text line will be visible. Here is how the **TEXTEDIT** feature works for the text that has been created using the **TEXT command**. If you had chosen text that was created using **MTEXT**, AutoCAD would have automatically opened the text editor for you.




**To insert the word "three" into the middle of the selected line, follow these steps.**

- Position the cursor precisely at the midpoint of the text located between "**These**" and "**lines**" and execute a click. You will notice a flashing cursor, which indicates the insertion point for typing new text.
- To achieve the desired result, input the number three and include a space to ensure that the line is correctly displayed as "**These three lines**."




- Press the **Enter key** twice to finalize the edits and exit the command. The cursor and highlighting vanish.

Now you will be able to make edits to the text you have created using **MTEXT**. One notable distinction is that when working with **MTEXT**, which generates multiple lines of text as a group, it is not possible to select an individual line of text. When an object is double-clicked or selected, all of its lines are automatically selected.

- Locate the text that starts with "**Mtext creates**" and double-click on it.

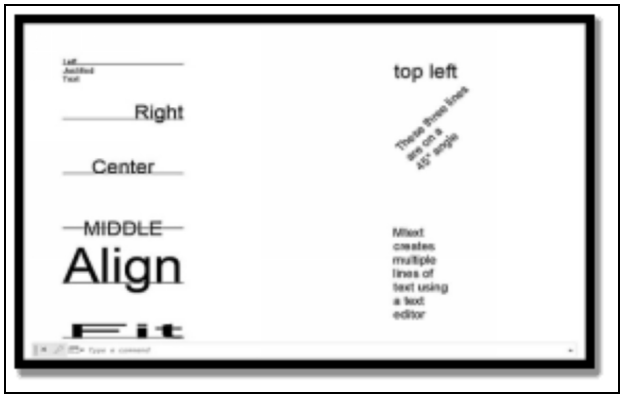
When you select the text that was created using **MTEXT**, **TEXTEDIT** will automatically open the Multiline Text In-place Editor. This editor displays the chosen paragraph within a text window. The Text Editor tab is also visible on the ribbon. When using the in-place text editor, the cursor will be automatically positioned in the line where you double-clicked.

- Click to the left of the word "**lines**" in the in-place text editor. When you open the program, a white cursor will be blinking at

the start of the line.

- Use the "**multiple**" command to create multiple lines of text using a text editor in AutoCAD. It is worth noting that "**Mtext**" is highlighted with red dashes. It seems that the in-place text editor is unable to recognize this word, suggesting a possible misspelling.
- To exit the **Multiline Text Editor**, simply move the cursor outside the in-place text editor and click.

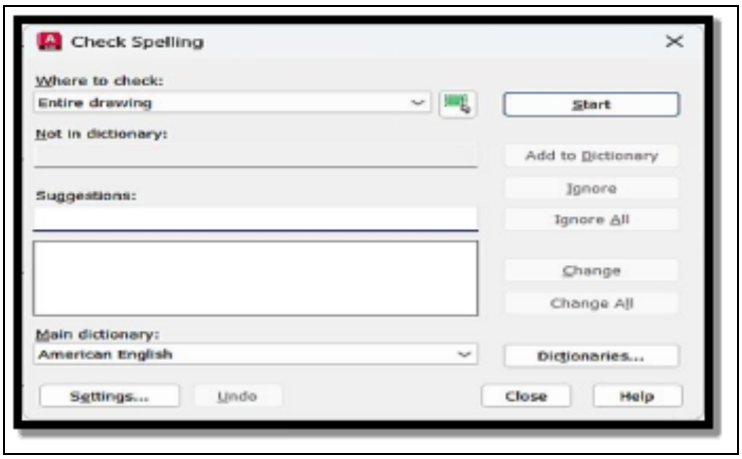
After returning to the drawing window, you will notice the newly added text. Additionally, you can delve into the Quick Properties palette and familiarize yourself with the various **mtext** justification options.




# How to Use Spell and Find Commands

The SPELL command in AutoCAD is user-friendly and will be recognizable to those who have experience with spell-checkers in word processing software. When using AutoCAD, you can use the **SPELL command** to ensure the accuracy of the text you have created.

- To check the spelling in AutoCAD, you can either type "**sp**" and press Enter, or navigate to the Annotate tab, then the Text panel, and click on the **Check Spelling tool** on the ribbon.




The drop-down list located at the top left of the dialog box is used to specify the location to check. You have three options to choose from: **Entire drawing, Current space/layout,** and **Selected objects**. For this exercise, use the Entire drawing option to thoroughly examine the spelling of all text objects in the drawing.

- If needed, choose "**Entire drawing**" from the drop-down menu labeled "**Where to check**".
- Press the **Start button**.

If you have completed the exercise without any spelling errors, you will notice that the Not in Dictionary box displays "**MTEXT**" and suggests "**TEXT**" as a correction. Before you finish using **SPELL**, take a moment to explore the available options. You can disregard any words that the checker does not recognize or make changes to them. You can modify a single occurrence of a word or make changes to all instances within the currently selected text. To modify the main dictionary in AutoCAD, you have the option to either add a new word or switch to a different dictionary. To access the desired options, locate and click on the Settings button located in the lower-left corner. From here, you will have the ability to customize the objects that will be checked for spelling and specify which types of words should be ignored. If you have selected the Settings button, simply click on Cancel to be directed back to the Check Spelling dialog box.

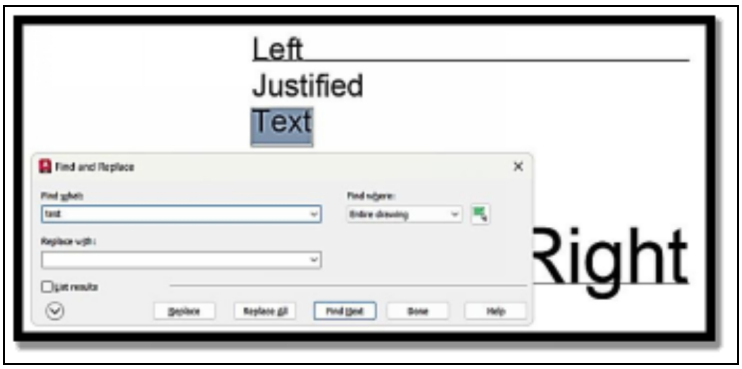
- Click the "**Ignore**" option. If your drawing is free of any other spelling irregularities, you should now observe a message from AutoCAD that says

**Spelling check complete.**

- Click the **OK button** to close the message box.
- To finish the **SPELL command**, simply click on the **Close button**. Any spelling corrections you have made will now be applied to your drawing.

The **FIND command** in AutoCAD enables users to locate specific elements or objects within a drawing. When it comes to this situation, the objective is to locate a specific text string within your drawing, rather than searching for your car keys or phone (although that would certainly be convenient, wouldn't it?). Once a text string is provided, AutoCAD will automatically zoom to the location of the first text object that matches the given string. This allows you to quickly understand the context in which the text appears. After locating a match, you have the option to either replace the text string with a new one or proceed to the next found match.

- To search for a specific text, simply type it in the Find Text box located on the Text panel of the Annotate tab on the ribbon. Press **Enter** to initiate the search.




The Find Where drop-down list, located at the top left of the box, offers the same options as the Where to Check drop-down list in the Check Spelling dialog box.

- Enter the desired text string into the Find what edit box to initiate the search.
- If needed, choose "**Entire drawing**" from the drop-down menu labeled "**Find where**".

- Locate and select the **Find button**.

When using AutoCAD, the software will automatically zoom to the first text object that matches the specified text string. Currently, you have the option to enter a value in the Replace with edit box and then click on Replace to substitute the existing text string with a new one. To replace all instances of a text string in the drawing, you have the option to click on "**Replace All**". This will automatically replace all occurrences of the specified text in the "**Find what**" edit box. When you click on the **Find Next option**, AutoCAD will take you to the next text object that contains the specified text string. To customize the search parameters and options, simply click on the **More Options button** located in the lower-left corner of the dialog box.

- Click the "**Done**" button to finish executing the **FIND command**. After closing the dialog box, you will be seamlessly redirected back to the drawing area.

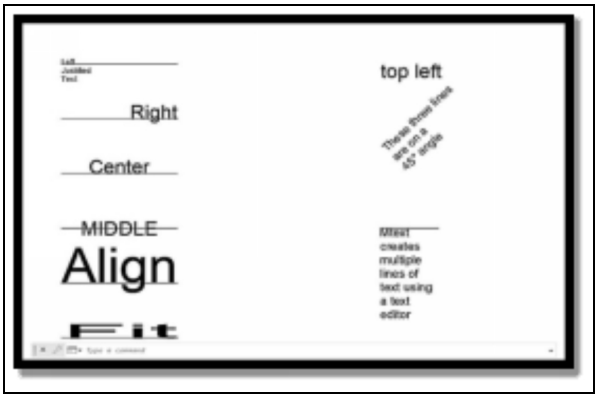
# How to Edit Text with the Quick Properties Palette

Every element within a drawing possesses unique attributes known as properties. Lines in AutoCAD have several standard properties that can be customized to meet your needs. These properties include **color, layer, line type, line weight,** and **length**. When you adjust these properties, you can create precise and visually appealing drawings in AutoCAD. Numerous additional properties vary depending on the type of object. When working with text in AutoCAD, you have several properties to consider. These properties include layer, content, justification, height, and rotation. By understanding and utilizing these properties effectively, you can create precise and visually appealing text elements in your designs. Additionally, there is a property called Annotative that will be further discussed in this book. In this exercise, you will discover the process of modifying the justification property of **mtext**. You can easily modify other properties by using the Quick Properties palette located on the status bar. Start by creating a visual reference line to ensure that any changes in justification are easily distinguishable. Properties are the essential characteristics of an object that

dictate its appearance and placement within a drawing. Certain properties are shared among all objects, such as a layer, color, line type, or line weight. However, some properties are specific to certain types of objects, like the radius of a circle or the endpoint of a line.

**Here's a helpful suggestion:**

- To access Quick Properties, you can simply select it from the Customization menu located on the status bar.
- Choose one or more objects.
- Locate and select the Quick Properties button located on the status bar.
- Use the Quick Properties palette to effortlessly modify property values.
- To close the Quick Properties palette, simply click on the close button located at the top right corner of the palette. Create a line that is 2.00 units long and positioned just above the mtext paragraph.



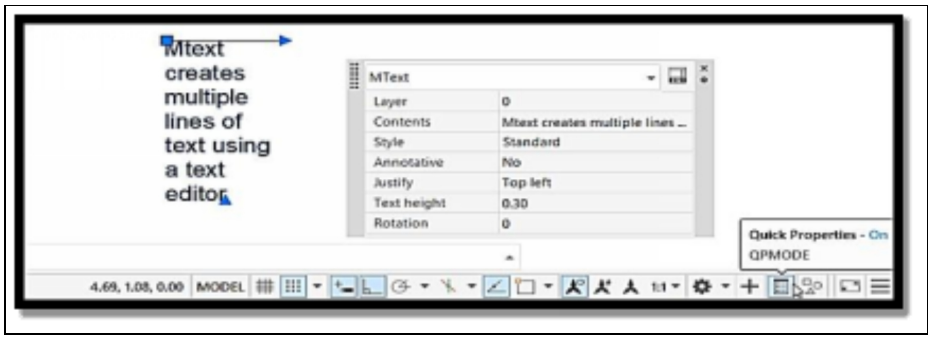

This line will provide a clearer understanding of the various justification options for placement. The starting point of the text should align with the snap point of the insertion point. The starting point of the line indicates the insertion point, while the length represents the width of the paragraph.

- To access the Quick Properties feature, locate the Customization button on the status bar and click on it. From the menu that appears, choose **Quick Properties**, which can be found towards the bottom.

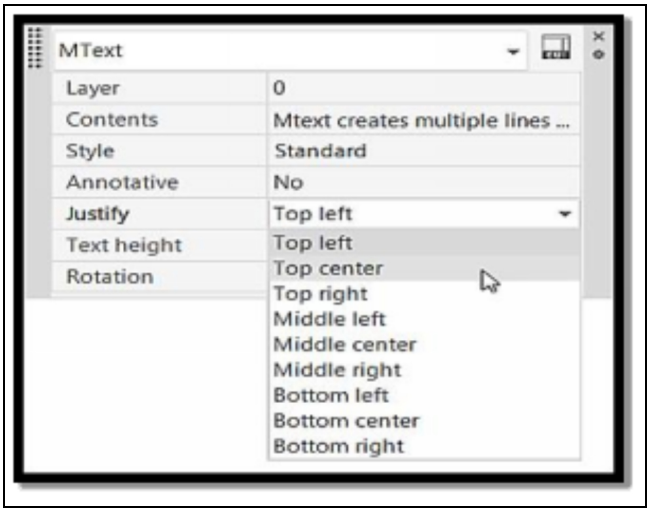
**By following these steps, you can easily add the Quick Properties button to the status bar.**

- Locate the mtext object containing the text "**Mtext creates**." The text is selected, with handles positioned at the top and bottom of the text window.
- Locate and select the **Quick Properties button** located on the status bar.

When the Quick Properties tool is enabled, you will be able to view the palette. The window is designed to float, allowing it to be positioned anywhere on your screen. To move it, simply click and drag the right or left border. The palette is displayed as a convenient two-column table, showcasing properties on the left and their corresponding current values on the right. Certain property values can be modified by entering text, similar to typing in an edit box, while others can be modified by selecting options from a drop-down list.

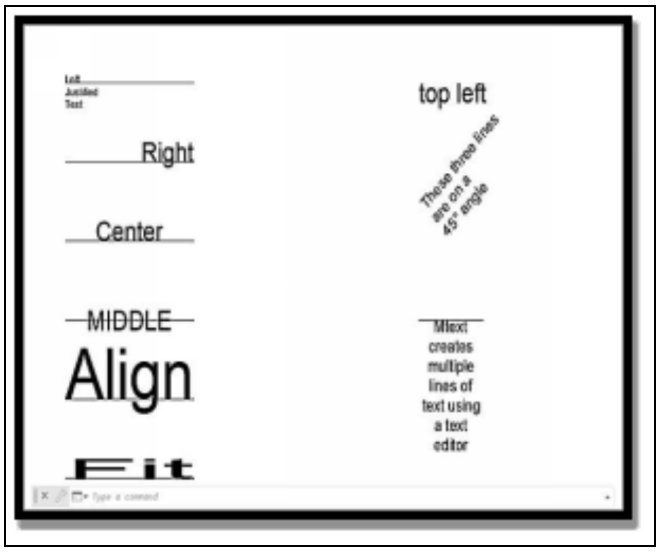



- Choose the value located at the top left of the column on the right, which is directly across from the **Justify property**.
- To access the drop-down list, simply click on the arrow located on the right side.

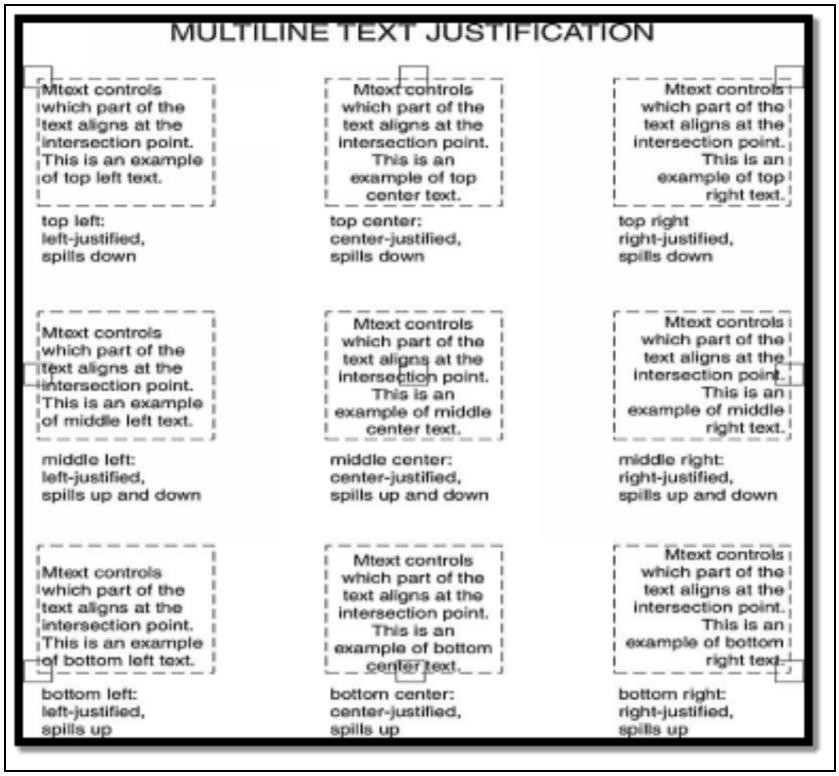



- Choose **Top Center**.
- Identify the top center on the list, but refrain from selecting it.

When you select this portion, the alignment of the mtext in your drawing is automatically adjusted. The paragraph is now displayed in a top-centered format. The in-canvas property preview feature is not limited to text justification but can be used with various object properties. It would be beneficial to explore additional justification options before proceeding.




**The text is centered.**




# Aligning multiline text in AutoCAD

The in-canvas property preview is a valuable feature that provides users with a visual representation of how a property change will affect their design before they make the change. This allows for better decision-making

and ensures that any modifications made are accurate and aligned with the desired outcome.

- Choose "**Top center**" to confirm this selection.
- To close the palette, simply locate the Quick Properties button on the status bar and give it a click.

To close the palette and keep it closed while you make further edits to your drawing, follow these steps. Before proceeding, it would be beneficial to examine the Quick Properties palette for a line.

- Use the <**Esc**> key to easily clear grips.
- Choose any of the lines in your drawing.
- To access the Quick Properties feature, simply locate and click on the **Quick Properties** button located on the status bar.

You will encounter a palette, ensure you examine the palette and the four properties displayed there.

- To close the palette, simply click on the Quick Properties button located on the status bar.
- Use the <**Esc**> key to easily clear grips.

# How to Change Properties with MATCHPROP

The **MATCHPROP command** is a highly efficient tool that allows you to effortlessly match the property values of one object to another object, whether it's all or just a selection. Some of the property values that can be transferred from one object to others are the **Layer, Linetype, Color,** and **Linetype scale**. These properties are universal across all AutoCAD entities.

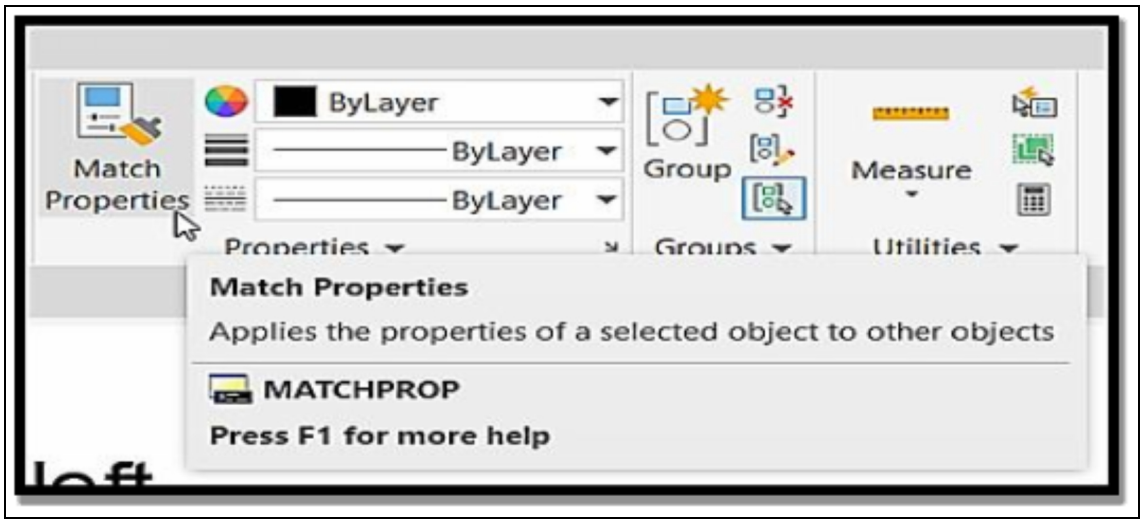
Thickness, Text Style, Dimension Style, and Hatch Style are additional properties that are specific to certain types of entities. Regardless of the situation, the process remains consistent.

**Here is a step-by-step guide on how to change property values using MATCHPROP:**

- To access the **Match Properties tool**, navigate to the **Properties panel** located on the Home tab of the ribbon.
- Choose a source object that has the desired property values to be transferred to another object.
- If needed, indicate the specific properties you wish to match using the Settings option.
- Choose the objects you want to move.
- Press the <**Enter**> key to conclude the selection of objects.
- To change the style of previously drawn text to the new Slanted style, you can use the **MATCHPROP command**.
- To access the Match Properties tool, navigate to the Properties panel on the Home tab of the ribbon.

**AutoCAD prompts:**
Select source object:
Multiple destination objects can be assigned, while only a single source object is allowed.

- Choose the "**Roman Duplex**" text that was created in the previous exercise, using the **Slanted style**.

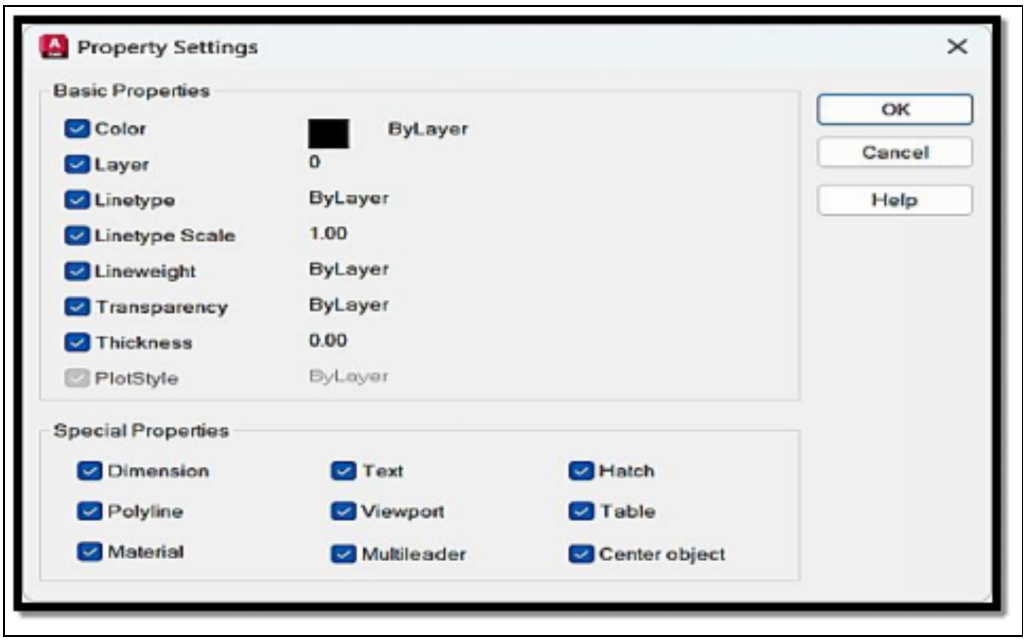



**Choose the destination object(s) or specify the desired settings:**

At this stage, you have the option to narrow down the properties you wish to match using the Settings feature. Alternatively, you can choose to select the destination objects, in which case all properties will be matched.

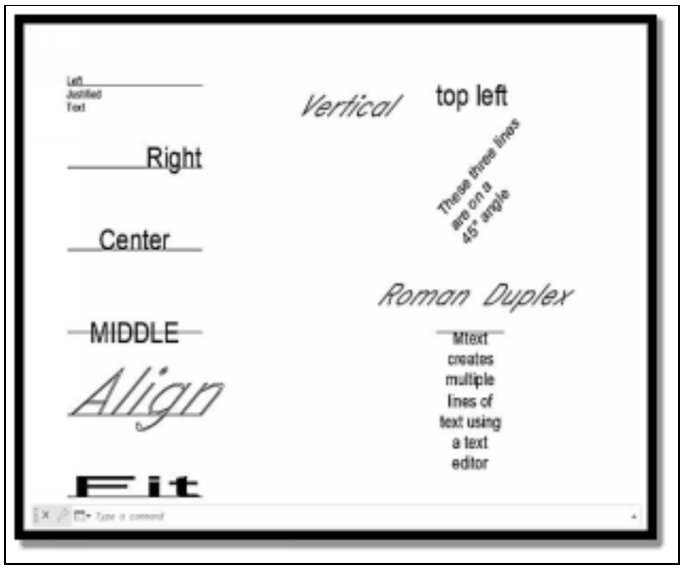
- Access the Settings menu using the command line.

By selecting this option, you will be able to access the Property Settings dialog box. The Basic Properties panel displays the modifiable properties and the corresponding settings that will be applied, depending on the selected source object.




Located at the bottom of the interface, there are nine additional properties displayed in the Special Properties panel. These properties and styles are defined within your drawing. When any of these options are not selected, **Match Properties** will disregard it and only match the selected properties.

- Click on the **OK button** to close the dialog box. When you open AutoCAD, you'll notice that it returns to the drawing area with the same prompt as before.
- Hover your cursor over any of the text in your drawing. AutoCAD provides a preview of how the text would appear in the Slanted text style.
- Identify the text objects containing the words "**Align**" and "**Vertical**" within your drawing. The Slanted style is applied to these two text objects. After executing a command in AutoCAD, the software will display the ***"Select destination object(s)"*** prompt, allowing you to proceed with selecting the desired objects.




# Align and slant vertical text

- Press the **Enter key** to exit the command.

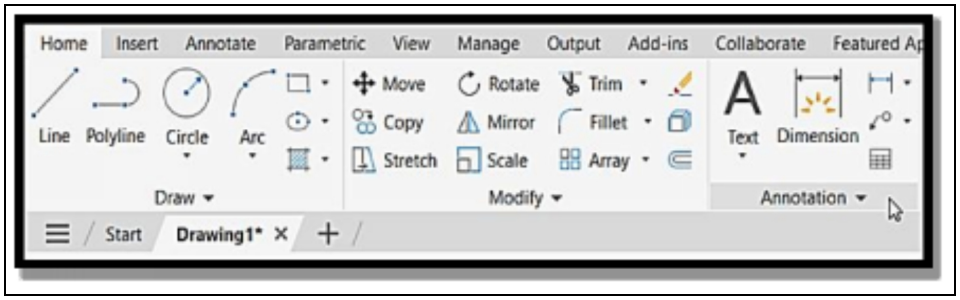
# How to Change Styles and Fonts

The default text style in any AutoCAD drawing is called Standard. The Arial true-type font is used, just like in a word processing program. All the text you have entered thus far has been drawn using the Standard text style, which is commonly used in AutoCAD. In AutoCAD, text and dimensions with the Annotative property can automatically scale to match the scale of viewports in a drawing layout.There is a style called **Annotative** that is pre-defined in all AutoCAD drawings. The **Annotative** style is identical to the Standard style in every aspect, except for the fact that it has the Annotative property enabled. This feature enables the creation of text that can be automatically adjusted to match the scale of various viewports within the same layout. Fonts are essential for creating different character and symbol shapes within the TEXT and **MTEXT commands**. Styles refer to the different variations in the size, orientation, and spacing of characters within fonts. Creating custom fonts can be a complex endeavor that may seem daunting to many. On the other hand, crafting your styles is simple and efficient. Text styles are used in conjunction with dimension, multileader, and table styles, which will be further elaborated on in subsequent chapters.

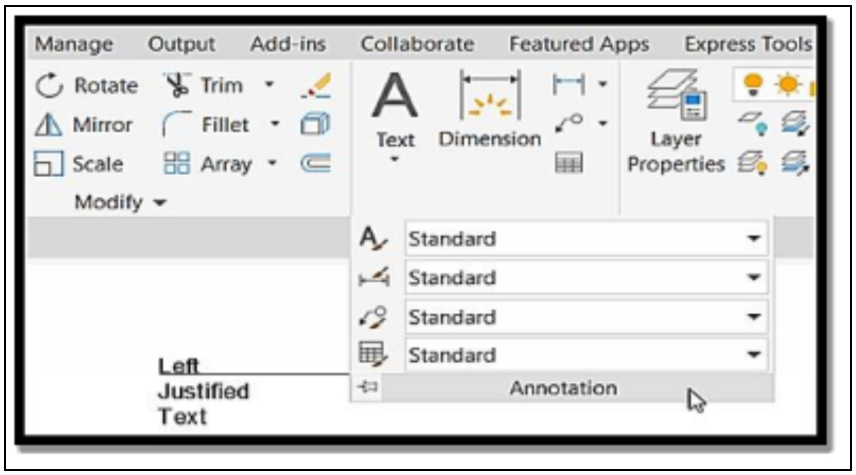
# Expanded Ribbon Panels

Start by creating a modified version of the Standard style that you've been using. This process will familiarize you with the panel slideout feature of the ribbon. It is important to observe the presence of a small downward-facing triangle on the right side of the majority of panel label names. To reveal additional tools, simply click anywhere along the label bar of a panel. For this next exercise, you will be using the tools found on the Annotation panel of the **Home tab**.

- To access the Annotation panel, navigate to the **Home tab** and click on the corresponding label.




When the cursor is moved away from the panel, it will close immediately. When the cursor is placed on the expanded area, the panel will stay open. To ensure that the panel remains expanded even when you navigate away, simply click on the pushpin located in the lower-left corner of the panel. To close it after being pinned, simply click the pushpin once more.

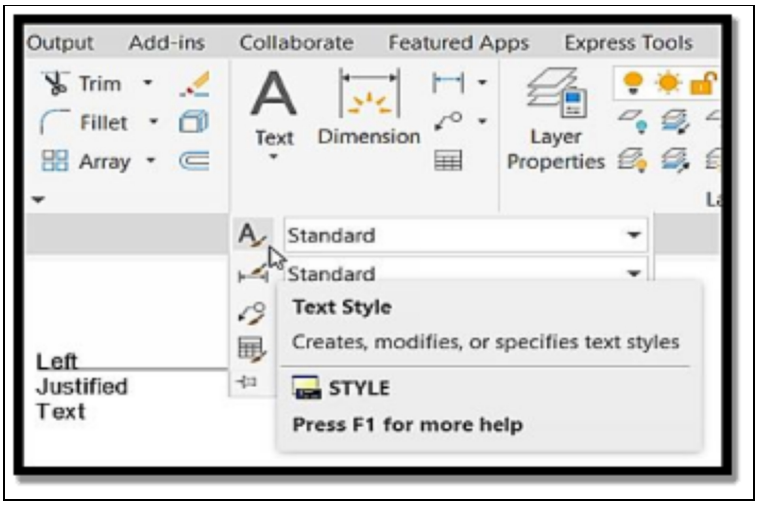


When you expand the **Annotation panel**, you will notice the presence of four drop-down lists. There are various lists available that provide access to different types of annotation styles. This chapter introduces text style, dimension style, and table style.

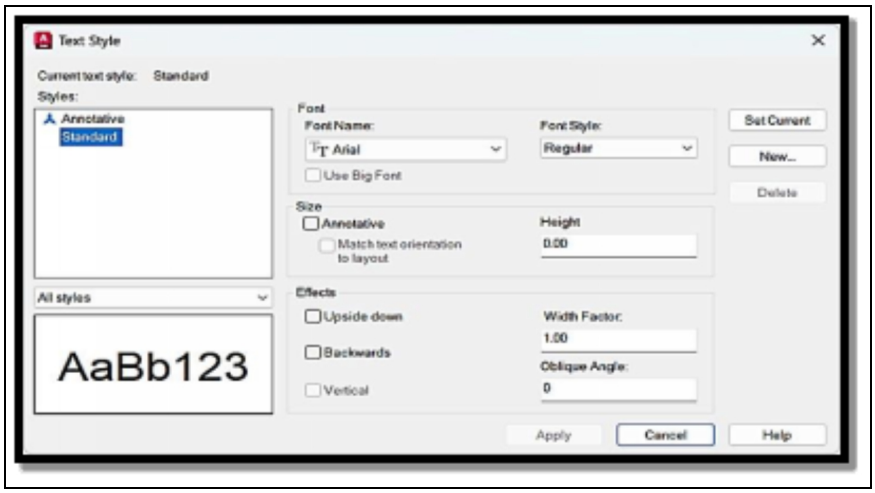
# NOTE

Additionally, you'll find a drop-down list for **Text Style** on the Text panel of the Annotate tab. It is important to have a clear understanding of the distinction between the Annotate tab and the **Annotation panel** on the **Home tab**. However, both of these locations provide access to text styles.

- Click on the **Text Style tool** in the Annotation panel.




In the Styles list box, you will find both Annotative and Standard options listed. There may be additional styles available. It is advisable to choose the standard option.




When creating a new style, it's important to select a font that differs from Arial. The new style will be called "**Vertical**" as it will be drawn in a downward direction rather than horizontally.

- Press the **New button**. When using AutoCAD, a smaller dialog box will appear, asking for the name of the new text style.
- Press the **Enter key** to create a vertical line. You will be directed back to the Text Style dialog box, where the newly created style will be displayed in the Styles list box. Afterward, you can assign the **txt.shx font** and apply the vertical text effect to give this style a distinct appearance. Unfortunately, the vertical text feature cannot be accessed if the Arial font is assigned to a style.
- Access the Font Name drop-down menu and locate **txt.shx** by scrolling through the options. There is a wide selection of fonts to choose from.
- Choose **txt.shx** to assign it as the font for the style. You can find the Vertical option in the Effects panel located in the lower middle of the dialog box.
- Enable the **Vertical option** by selecting the corresponding check box in the Effects panel.

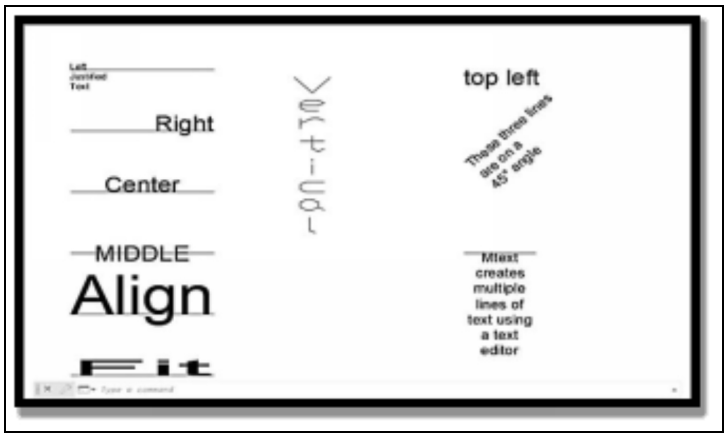
Take note of the alteration to the text orientation in the Preview panel located at the bottom left of the dialog box. Subsequently, you can assign a specific height and width factor to this style. Take note of the current height, which is currently set at **0.00**. Rest assured, the height of your characters will not be set to 0.00 units. This indicates that the height will not

be predetermined, allowing you to designate a height whenever you utilize this style. The standard height is currently not fixed, so the vertical setting has inherited this configuration.

- Simply double-click on the Height edit box and enter the value of 0.5.
- Simply double-click on the Width Factor edit box and enter the value of 2.

The current text style should be automatically set as vertical. To make it current, simply highlight it in the Styles list box and click the **Set Current button** on the right.

- To make Vertical the current text style, click the **Set Current button** if needed. If a message box appears indicating changes to the text style, simply click ***Yes*** to save the changes.
- To exit the Text Style dialog box, simply click on the **Close button**. The Vertical style has been updated and is now active. To witness its functionality, you will be required to input a certain amount of text.
- Access the **Single Line Text command** by navigating to the **Home tab** on the ribbon, then selecting **Annotation** followed by Text.
- Specify the starting point.



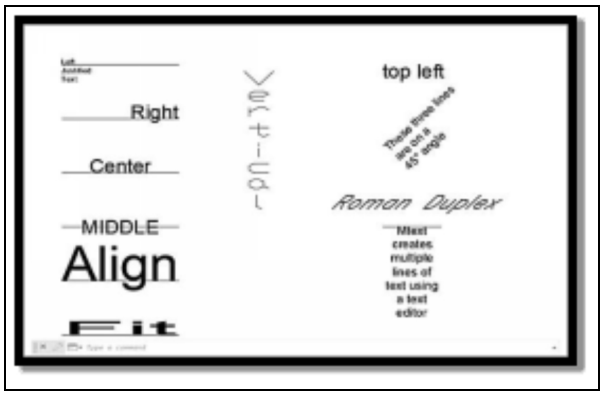

It is important to note that there is no prompt for a height in this case, as the current style has a fixed height of 0.50. It is important to be aware that the

default rotation angle is set to 270. The vertical text is entered in a downward direction at an angle of **270**°.

- Press the **Enter key** to maintain a rotation of 270 degrees.
- Press the **Enter key** after typing "**Vertical**".
- Press the **Enter key** once more to conclude the **TEXT command**.

Now, you can proceed to create another style by selecting a different font and exploring the various style options available. Keep an eye on the **Preview panel** in the dialog box as it will automatically update to reflect any changes you make.

- In the Home tab, you can find the Annotation panel. Expand it and locate the **Text Style** tool. Click on it to proceed.
- To begin, locate and click on the "**New**" button.
- When you open the New Text Style dialog box, go ahead and give the style a name like "**Slanted**".
- Click the **OK button** to close the New Text Style box.
- Access the Font Name drop-down list by selecting the arrow.
- Locate the font file named **romand.shx** and come to a halt. Romand is an AutoCAD font known as Roman Duplex.
- Choose the **romand.shx font**.
- Disable the **Vertical checkbox**.
- Adjust the text height to **0.00**.
- Adjust the **Width Factor** to a value of 1.
- Adjust the **oblique angle** to **45 degrees**. By applying this technique, your text will be tilted at a 45° angle toward the right. To achieve a left slant, input a negative value.
- To exit the **Text Style dialog box**, simply click on the **Close button**.
- Click on the "**Yes**" button to save the changes you have made. Input some text to observe the appearance of the Slanted style.
- Use the Single Line text tool and follow the prompts to create the text "**Roman Duplex**" with a height of 0.50 and no rotation.




**Take note:** When modifying a text style, it's important to note that any text created using that style will be automatically updated to reflect the new specifications.

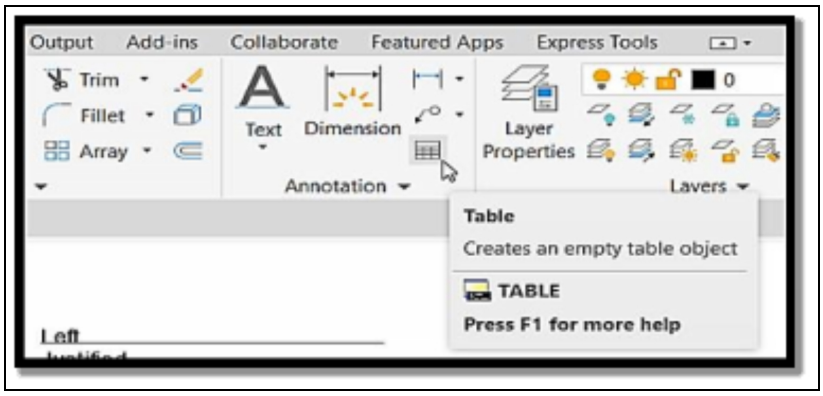
# Switching the Current Style

Text in AutoCAD is generated using the existing style. You can modify the appearance of text that has already been created. After defining multiple styles in a drawing, effortlessly switch between them by using the Style option within the **TEXT** and **MTEXT** commands or by selecting a text style from the convenient Text Style drop-down list. To access the Annotation panel, simply expand it by clicking on it. Then, locate the arrow on the right side of the text style and click on it. From the drop-down list that appears, choose the desired style for your text.

# How to Create Tables and Fields

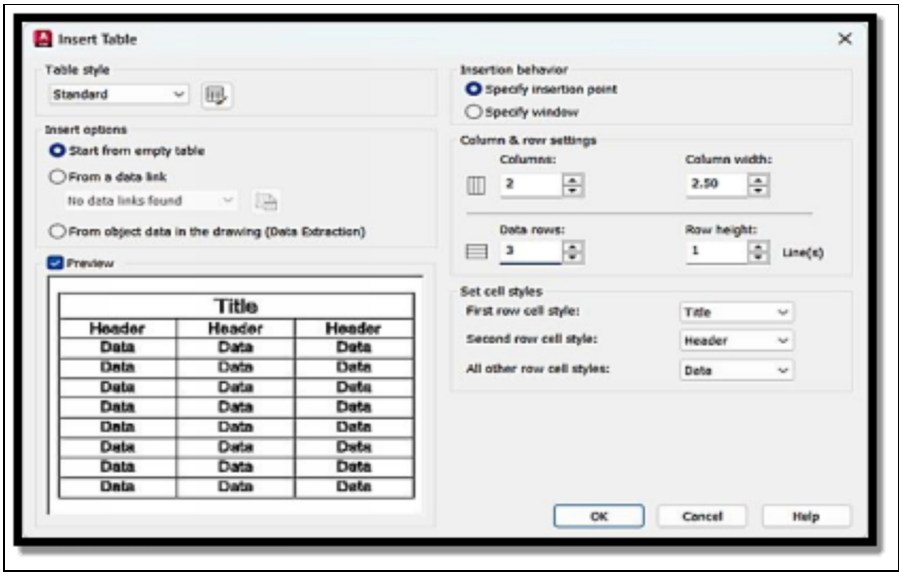
AutoCAD offers a wide range of powerful tools and functions to efficiently create and handle tables and their associated data. You can create and assign names to table styles in AutoCAD. You can format cells in AutoCAD to include text, numerical data, formulas, and fields that will update automatically. You can easily extract data from the current drawing and organize it into a table or import it from an external spreadsheet. In this exercise, you will learn how to create a basic table with three different date and time formats and have complete control over their appearances. The table will consist of a title, two columns with headers, and three rows of data. Furthermore, you gain knowledge about fields, which can be effortlessly inserted and updated whenever there are changes to the information they contain.

- For this exercise, you can start with any AutoCAD drawing. For optimal use, refer back to the drawing containing the previously created text objects from the previous chapter. Zoom in to the specific area where the table is intended to be inserted.
- To zoom in, locate an empty area in your drawing and adjust the view to a size of approximately 5.00 by 5.00.
- To access the **Table** tool, navigate to the **Annotation** panel on the Home tab of the ribbon. Locate the tool.




# Table tool

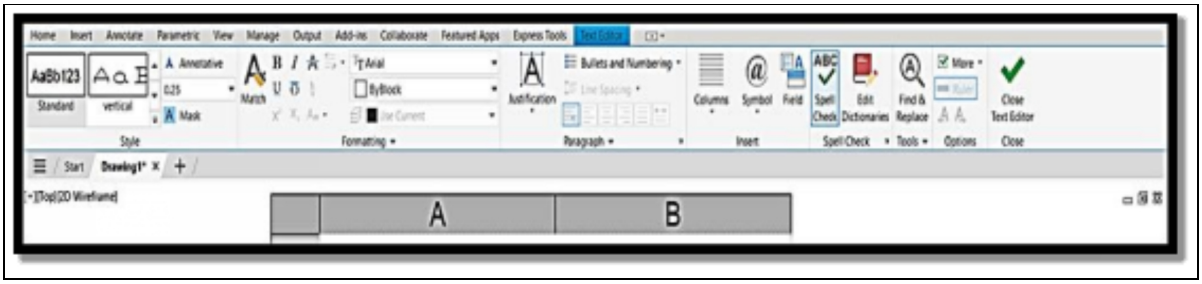
You can find more tools for working with tables and table data on the Tables panel of the Annotate tab. When you perform this action, the **Insert Table dialog box** will appear. Before inserting a table, you would need to define its dimensions, form, and design. The **Standard** table style includes a title row, a header row, and multiple rows and columns of data. The table you insert will utilize the Standard style and consist of three rows and two columns.




- Ensure that the Table style drop-down list includes the option for "**Standard**". Make sure that the Insertion behavior panel is set to "**Specify insertion point**", and that the Insert options panel is set to "**Start from empty table**".
- In the Column & row settings panel, you can easily specify 2 columns by using the arrows or the edit box.
- Adjust the column width to **2.50** if needed.
- Use the directional arrows or the designated input field to accurately indicate the desired number of rows, which in this case is 3.
- Adjust the row height to a single line if needed.
- Click the **OK button**. After closing the dialog box, AutoCAD conveniently displays a table that seamlessly follows your crosshairs as you effortlessly drag it into the desired position.
- Specify the desired location for insertion. You have successfully created a blank table within your drawing. The table consists of a title cell at the top row, followed by four rows and two columns of empty data cells. The numerical and alphabetical labels on the left and top of the table serve as references for rows and columns, but they do not contribute to the final table itself. The text cursor should be flashing in the title cell, ready for text input. It is important to note that the **Text Editor tab** is displayed and currently selected on the ribbon. First, enter a title and then proceed to input the two-column headers. For optimal

consistency, it is important to begin by specifying the desired text height.

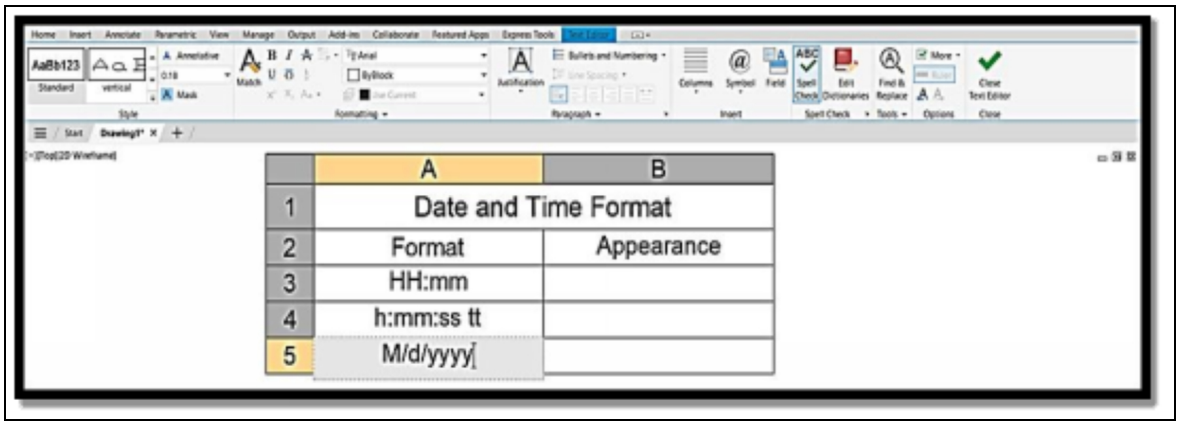
- Access the Text Height drop-down list box on the Style panel of the **Text Editor tab** by double-clicking on it. Then, enter the value **0.20** and press **Enter**.




**The current text height has been set to 0.20.**

- To set the title, enter "**Date and Time Format**" and press **Enter**. After pressing <**Enter**>, the text cursor will automatically move to row 2, column A. The cursor is now blinking in the 2A cell, ready for input. Cell 2A is formatted as a header with a center justification for text.
- Enter "**Format**" in the first column and press the <**Tab**> key. Pressing the Tab key will navigate you to the subsequent header cell.
- To modify the appearance, simply press the "**Tab**" key after typing "**Appearance**".

Tabbing in AutoCAD allows you to seamlessly navigate to the next table cell, automatically wrapping to the next row when necessary. You are currently positioned in the third row of column A, which happens to be the first data row. Observe that the column headings are perfectly centered within their respective cells. The center justification for the column headers is a fundamental aspect of the **Standard** table style.



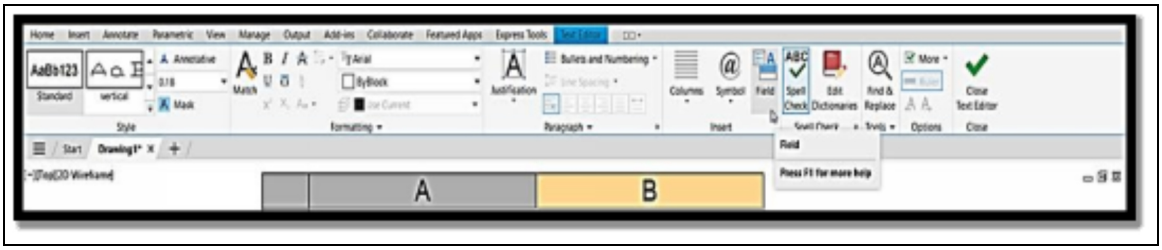

**Here's a helpful suggestion:** To navigate through table cells sequentially, you can use the <**Tab**> key. To reverse the direction and move backward, simply hold down the **Shift key** and press **Tab**. Additionally, the arrow keys on your keyboard can be used to navigate between table cells. When inputting data into the table, you'll need to specify the date and time format definitions in the Format column, while the corresponding date and time fields should be entered in the **Appearance column**. The information in the **Appearance** column can be inserted as fields that can be updated automatically. In this exercise, dates and times are used due to their high level of responsiveness and ease of updating.

- Enter "**HH:mm**" in the first column of the data row, and press **Enter**. This symbol represents time in the standard hours and minutes format. When the uppercase HH is used, it signifies the representation of time in the 24-hour format. For example, 2:00 P.M. will be displayed as 14:00. When you press the **Enter key**, it will navigate you to row 4, column A.
- Enter the time in the format **h:mm:ss tt**. This represents time in hours, minutes, and seconds. The abbreviation "**tt**" is commonly used to represent "**A.M.**" or **PM**. You'll be seeing more of these symbols shortly.
- Enter the date in cell 5A using the format **M/d/yyyy** and then press the **Tab key** once. This is a date expressed in the format of month, day, and year, with a four-digit number for the year.

# Adding Fields

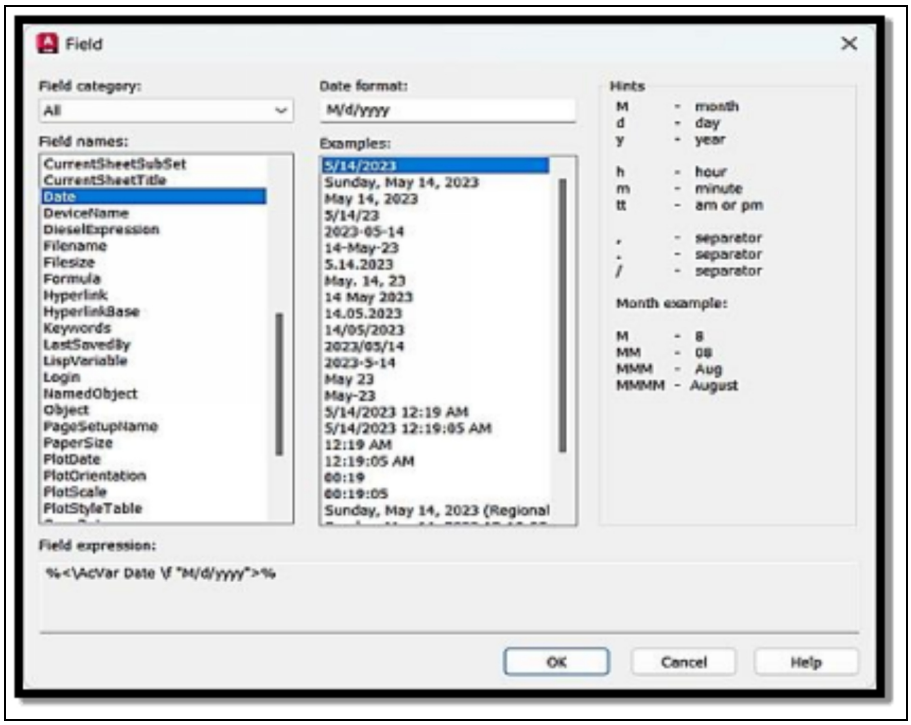
You are currently located in cell 5B of the table. Instead of entering text in this cell, you can insert a date field in the format of month, day, or year to align with the **Format column**. Next, you can proceed to the remaining rows in this column and input the time fields.

- Click on the **Field tool** located within the **Text Editor tab** on the **Insert panel** of the ribbon.



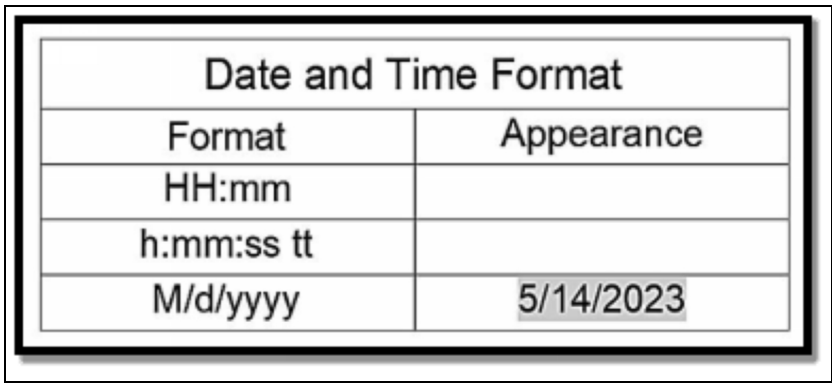

# Use the Insert Field tool

By selecting this option, you will be able to access the Field dialog box. Within the Field names list box on the left, you will find a wide array of predefined fields to choose from. The content displayed in the Examples list box varies depending on the type of field that is chosen.




- Choose the desired date from the list of field names. When the **Date option** is selected, you will be able to view various example formats for dates and times. There is a wide range of options available, requiring you to scroll to view them all. Observe that the date format symbols for the chosen format are shown in the **Date format** edit box above the examples.
- Choose the example located at the top of the list to display the current date in the **M/d/yyyy format**.
- After selecting the desired format, simply click **OK**.

*The date is entered in the cell. The display color of this element is gray, which signifies that it is a field and not regular text. If this drawing were to be plotted, the gray color would not be visible in the plot*.




Enter the date in the desired cell. Now, it's time to exit the table and go back to the drawing.

**NOTE**

Fields can be inserted at any location within a drawing. To insert a field without a table, you can easily access the Field tool by navigating to the **Data panel** on the **Insert tab** of the ribbon. To open the Field dialog box, follow the same steps as before.

- To navigate back to the drawing, simply click outside of the table.

It is possible to fill out all the table cells without exiting the table. However, it is crucial to understand how to input data after creating a table. The selection of cells can vary depending on the sequence used. Clicking a cell will select it, causing the **Table Cell tab** to be displayed on the ribbon. When you double-click within a cell, the **Text Editor tab** will be displayed on the ribbon.

- Click on the cell located in the first row of data, second column, right below the **Appearance** header. The cell is selected, and the Table Cell tab is displayed on the ribbon.
- To access the Field tool, navigate to the **Table Cell tab** and locate it in the Insert panel of the ribbon.

When you open the Field dialog box, you will notice that the Date option is already selected in the Field names list box. Choose the **HH:mm** example to align with the format specified in the Format column. This particular selection can be found as the seventh option from the bottom in the

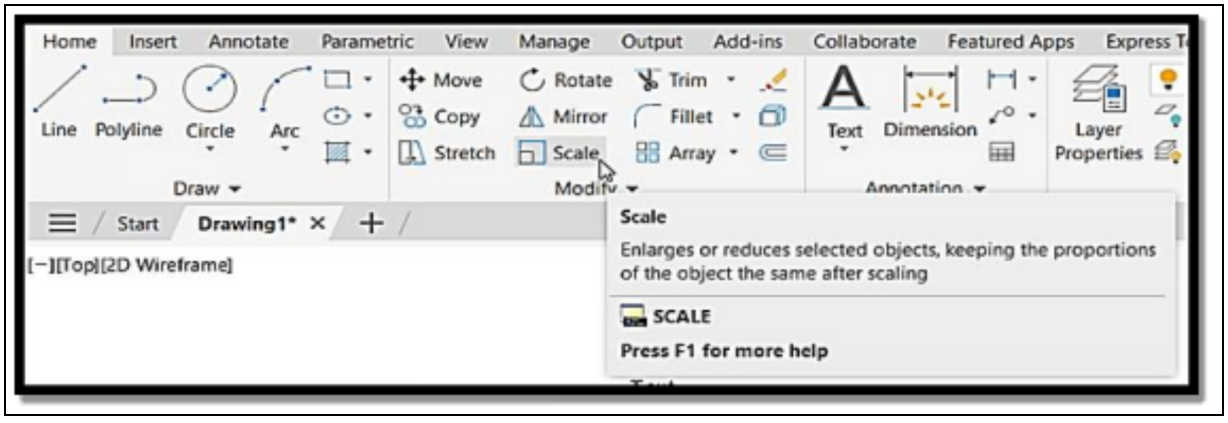
Examples list box. The time will be displayed in a 24-hour format, showing hours and minutes without any indication of A.M. or PM. Job title.

- To locate the **HH:mm date format** example, simply scroll down to the seventh entry from the bottom of the list. When the Date format edit box is highlighted, the time format **HH:mm** will appear.
- Click the **OK button**. Lastly, you can repeat the steps to insert a field in the remaining cell.
- Use the downward arrow key on your keyboard to navigate to the next row.
- Click on the **Field tool** once more.
- Choose the **h:mm:ss tt format** from the list of examples. This format is located right above the **HH:mm date format**.
- Click the **OK button**.
- Use the <**Esc**> key to easily clear the grips of the selected cell.

# How to Scale Previously Drawn Entries

You can easily adjust the size of any object or group of objects by using the **SCALE command** or the Scale grip edit mode. In this exercise, you will learn how to scale text and lines in AutoCAD. This is an important skill to have when working with drawings. It's important to note that the **SCALE command** in AutoCAD doesn't have any specific connection to text. You can scale other types of entities just as effortlessly.

- To access the **Scale tool**, navigate to the **Modify panel** on the Home tab of the ribbon.

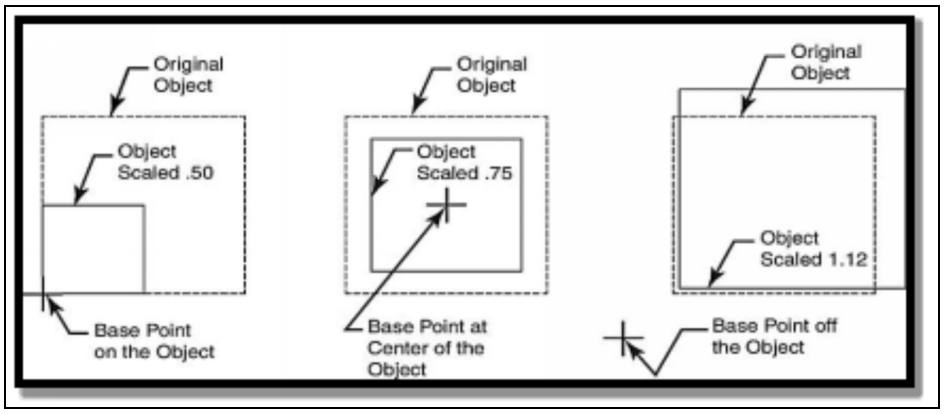


# The scale tool

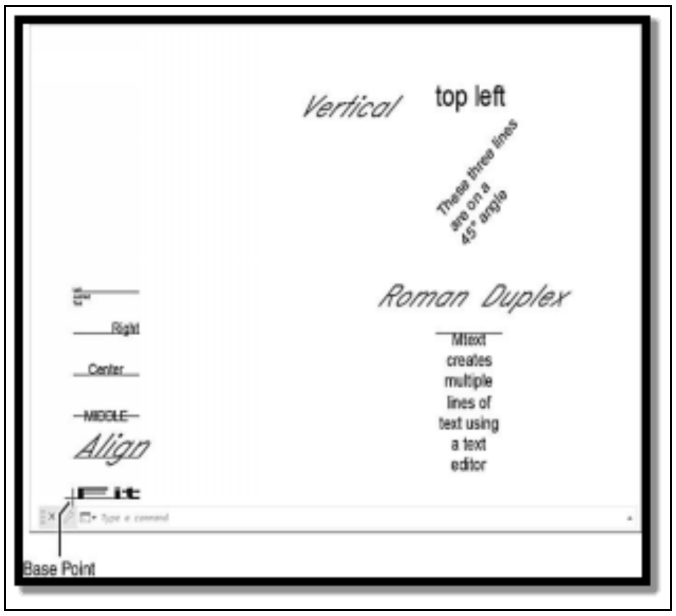
When using AutoCAD, the software will prompt you to select the desired objects. **Here is a step-by-step guide on how to change the scale of objects:**

- Access the **Scale tool** by navigating to the **Modify panel** on the Home tab of the ribbon.
- Choose the desired objects.
- Indicate the desired base point.
- Provide the scale factor.
- To select the set of six lines and text drawn, use a crossing window by dragging from right to left.
- To end the selection of objects, simply perform a right-click.

**A base point is required for input:**
**Specify base point:**
Understanding the importance of a base point in scaling is essential. Picture this scenario: you're observing a square and you're aiming to reduce its size using a scale-down technique. Uniformly reducing the size of all sides is straightforward, but could you please specify the desired method? It is important to keep the lower-left corner fixed while shrinking the entire square towards it. Should everything be scaled towards the center? Or perhaps to another location within or outside the square? When defining a base point, it is important to specify the details accurately.




- Specify the base point by selecting the left end of the bottom line of the objects you have chosen.




When working with AutoCAD, it is important to specify the desired scale for the selected objects to accurately shrink or enlarge them.
Specify scale factor or (Copy/Reference) <1,00>:
With the **Copy option** in AutoCAD, you can keep the selected objects at their original scale while also generating copied objects at a different scale. Scaling objects by reference will be discussed shortly. After inputting a scale factor, the selected objects' lengths, heights, and diameters are multiplied by the factor and subsequently redrawn to reflect the changes. The scale factors are derived from a unit of 1. When a value of 0.5 is entered, the objects will be scaled down to half of their original size. When the value of 2 is entered, the objects will increase in size by a factor of two.

- Enter **.5** and press the **Enter key**.

**Here's a helpful suggestion:**
Text objects can be scaled using the SCALE command. However, AutoCAD also provides a dedicated command specifically designed for resizing text objects. The **SCALETEXT** command allows you to scale selected text objects from their insertion points, without moving the text object during scaling.

# Scaling by Reference

This feature eliminates the need for manual calculations to determine scale factors, saving you valuable time and effort. Knowing the desired length

after scaling can be quite beneficial when you already have a given length. As an AutoCAD expert, it is evident that the lines you recently scaled have a length of 2.00. This is because they were initially drawn at a length of 4 and then scaled down by a factor of 0.5. If you want to adjust the scale of the objects to a length of 2.33, follow these steps: Calculating the scale factor can be a bit of a hassle, but fortunately, there's a simple solution. In this case, the scale factor is 1.165, which is obtained by dividing 2.33 by 2.00. No need to waste time on manual calculations.

**Here is the step-by-step procedure:**

- Open the command prompt and type in "**SCALE**" to access the scaling functionality.
- Reselect the line and text objects.
- Specify the base point.
- To access the Reference command, you can either press the "**R**" key followed by the **Enter key**, or you can right-click and choose "**Reference**" from the shortcut menu.
- Enter "**2**" to specify the reference length.
- Enter **2.33** as the new length.

Reference scaling can be achieved by specifying two points and utilizing the distance between them. In the given procedure, you have the option to define the endpoints of a 2.00-unit line as the reference length, and then specify the two endpoints of a 2.33-unit line as the new length.

# Scaling with Grips

**Scaling using grips is essentially the same as scaling using the SCALE command. To demonstrate this, use grips to revert the text you recently scaled back to its initial size.**

- To select the six lines, use a window or crossing window.

Multiple grips are visible in the drawing, with three on each line and two on most of the text entities. There are instances where certain elements are overlapping each other.

- Select the grip located at the lower-left corner of the word "**Fit**". This is the same point that was used as the base point in the

previous scaling procedure.

- To scale an object, simply right-click on it and choose the **Scale option** from the shortcut menu.
- Ensure that the cursor is moved at a controlled pace while carefully observing the image being dragged. In AutoCAD, the selected grip point serves as the base point for scaling, unless you explicitly indicate otherwise. It is important to note that there is also a **Reference option** available, similar to the **SCALE command**. In SCALE, the default method is to specify a scale factor by specifying points or typing.
- Enter the number 2 or indicate two points that are exactly 2.00 units apart. (The objects were previously scaled down by a factor of 0.5, so they need to be scaled up by a factor of 2 to restore them to their original size.)
- Use the <**Esc**> key to easily clear grips.

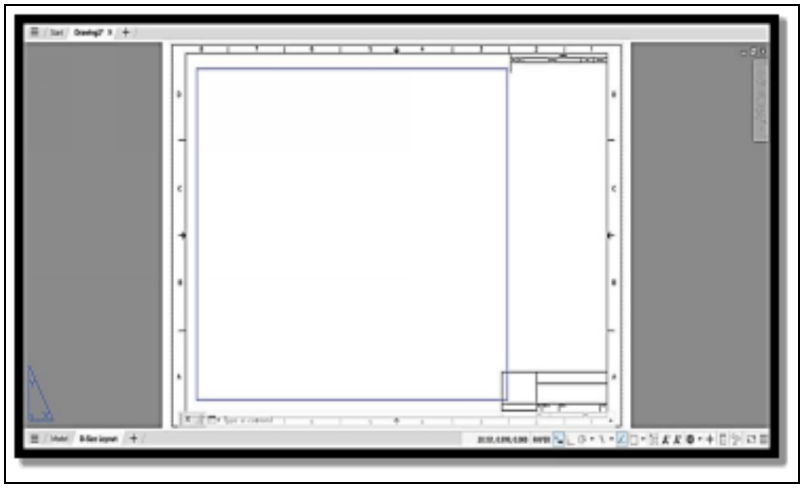
# How to Use Drawing Templates, Title Blocks, and Borders

Having a deep understanding of text creation and paper space layouts allows you to fully use the drawing templates that come with pre-designed borders and title blocks. In this exercise, you will learn how to create a new drawing using a D-size drawing template. You will also discover how to add simple geometry and incorporate text into the title block. Your current progress can serve as a foundation for finishing, specifically the gauges.

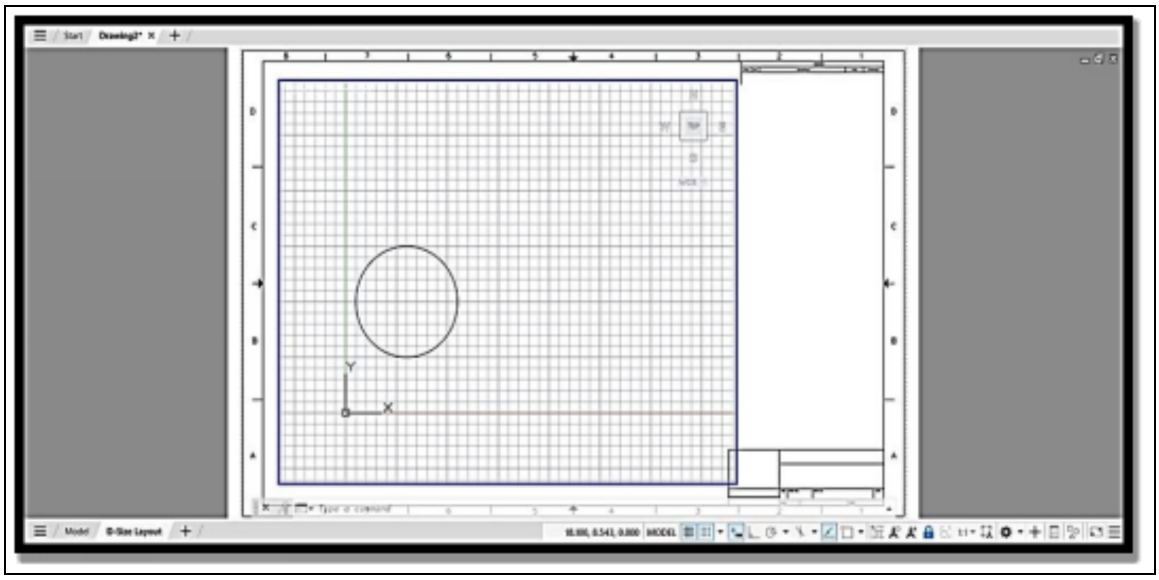
- To begin, locate and select the **New tool** located on the Quick Access toolbar.
- Choose the **Tutorial-iMfg template** from the **Select Template dialog box**.
- Enter "**Tutorial-iMfg**" in the File name box and then click **Open**.

When you open a new drawing layout, you will be greeted with a pre-drawn border and title block. Take note of the paper space icon located in the lower-left corner of the drawing window, as well as the Paper button that has now replaced the Model button on the status bar. Additionally, take note

of the window featuring blue borders that delineate a model space viewport. After accessing the viewport, proceed by creating a circle to initiate the drawing's geometry.



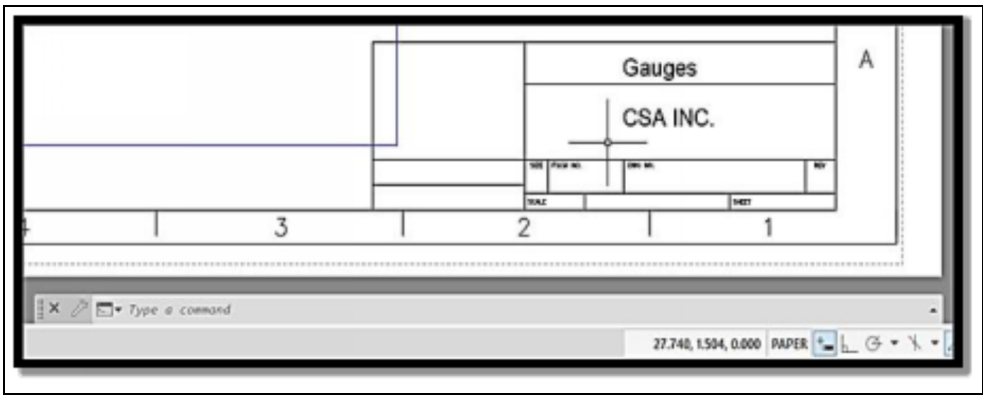
- To enter model space in this viewport, simply double-click inside the blue borders. The line weights increase. Enabling the grid provides a more comprehensive understanding of the drawing area within this specific viewport.
- Enable the grid within this viewport.
- To ensure the grid fills the viewport, you can access the Snap Settings by right-clicking the **Snap Mode button** on the status bar. From there, select the option to Display the grid beyond Limits in the dialog box.
- Now, let's proceed with the **CIRCLE command**.
- Create a circle with a radius of **2.50** and a center point at (**3.00,5.00**).



**To create a circle, follow these steps:**
When performing this task, it's important to observe that if you began this drawing using a different template, you will have a completely different arrangement of layers, and the units will be displayed in three-place decimal format.

- Disable the grid within the viewport. It is advisable to hide the model space grid before switching back to the paper space layout.
- Click twice outside the blue border. The paper space icon has made a comeback, and the viewport border has been adjusted to a more subtle style. Welcome to paper space. Now, you can easily incorporate two pieces of text into the title block. To streamline these entries, you can switch to the paper space snap.

The current snap setting in this template is set to .500, but you won't be able to see it until you enter a drawing command. Take note of the layout, which replicates the dimensions of a 33.000 × 21.000 D-size drawing sheet. To verify this, simply position your cursor at the corners of the effective area. However, it is important to note that you won't be able to snap to these points precisely unless you enter a **draw command**. Next, you can easily add text to two different areas of the title block. Unfortunately, the current snap setting is causing some difficulty.



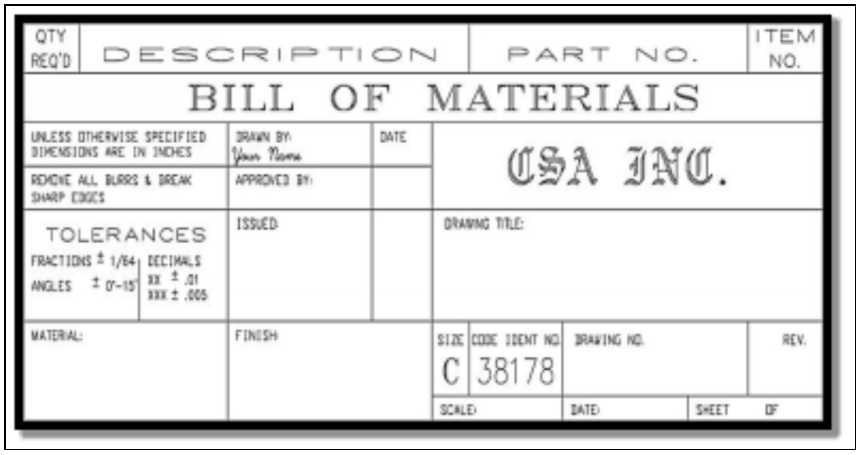

# Inserting text

- Press the **Enter key** after typing "**snap**".
- Choose the "**On**" option from the command line.
- Press the <**Enter**> key to repeat the **SNAP command**.
- To modify the snap setting, simply enter ".**250**" and press **Enter**.
- Press the "**z**" key and then use a window to zoom in on the title block. Now you're all set to add text.
- To access the Single Line text tool, navigate to the **Text drop-down menu** located on the Annotation panel of the Home tab on the ribbon.
- Specify the desired starting point near the coordinates (**28.000,2.500**).
- Set the text height to **.25** and the rotation angle to **0**.
- Enter "**Gauges**" and press the **Enter ke**y.
- Press the **Enter key** once more to exit **TEXT**.
- Repeat the text and input CSA INC. at coordinates (**28.000,1.750**).
- Press the **Enter key** twice to exit **TEXT**.
- To view the complete layout, simply type **'z'** followed by the **'Enter'** key, and then type **'a'** followed by the **'Enter'** key.

# Chapter Drawing Projects

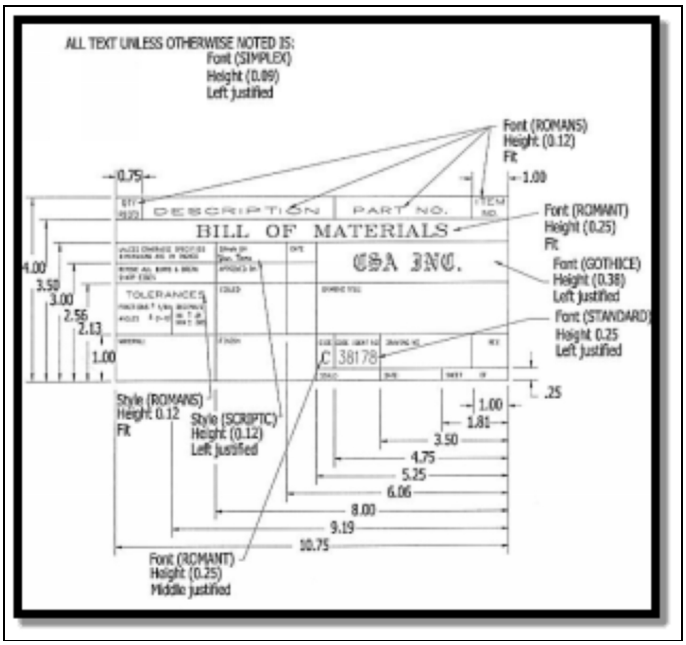
# Drawing 7-1: Title Block [INTERMEDIATE]

This title block provides an opportunity to enhance your skills in utilizing a diverse range of text styles and sizes. It would be advisable to save the file and utilize it as a title block for any upcoming drawings.



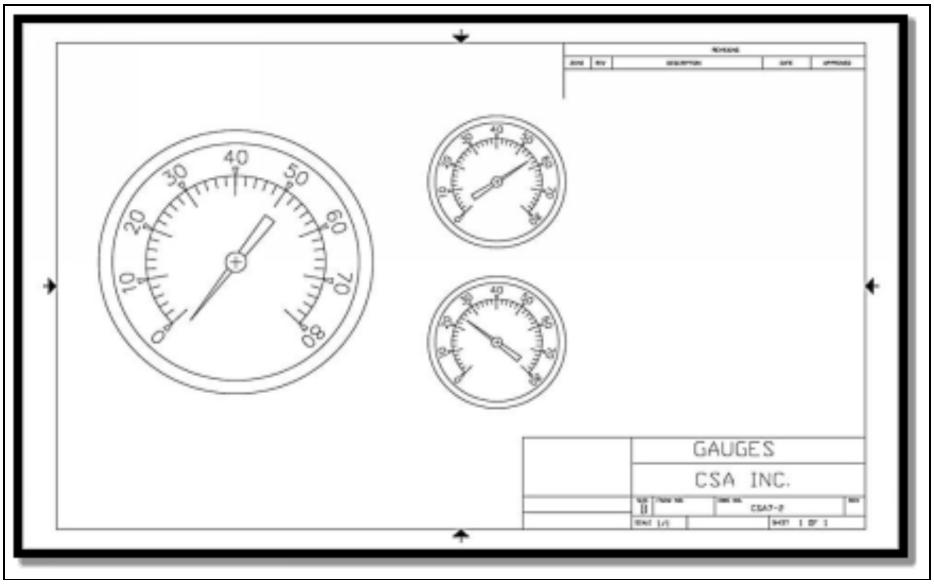

**Here are some suggestions for your drawings:**

Enable the grid with a spacing of 1 unit and disable the snap function with a spacing of 0.0625 units. Use the **TRIM command** extensively when creating the line patterns for the title block. It is important to carefully plan and establish the main divisions before proceeding to input text into the designated boxes. Ensure that the text layer is selected before creating new text. When creating text in **AutoCAD**, it is recommended to use the TEXT command. This command allows you to easily create standard, 0.09-size text that is left-justified. It is important to note that after defining a style, you can set it as the current style within the **TEXT** command. This prevents the need for excessive restyling. When working in AutoCAD, you can use %%D to represent the degree symbol and %%P to indicate the plus or minus symbol.




# Drawing: Gauges [INTERMEDIATE]

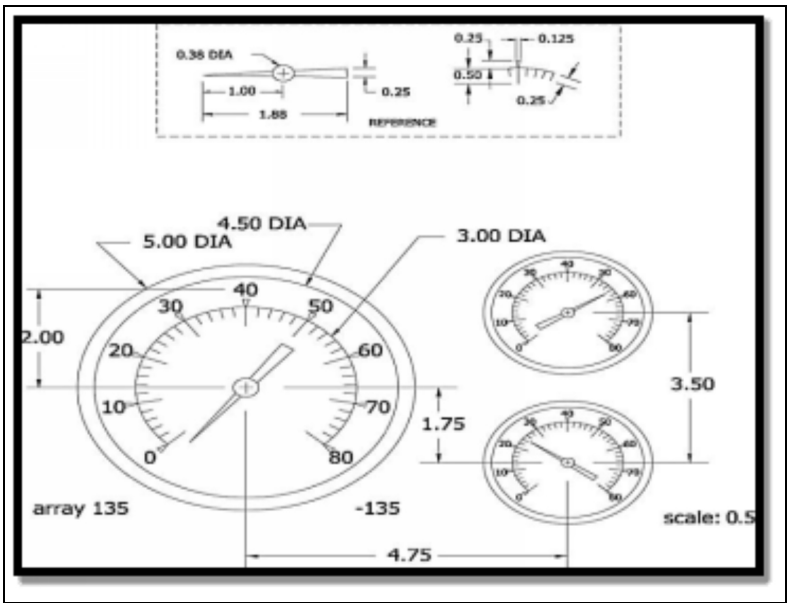
This drawing demonstrates common applications of the **SCALE** and **TEXTEDIT** commands. It is important to carefully review the suggestions as some of the techniques used may not be immediately apparent.




**Here are some suggestions for your drawings:**

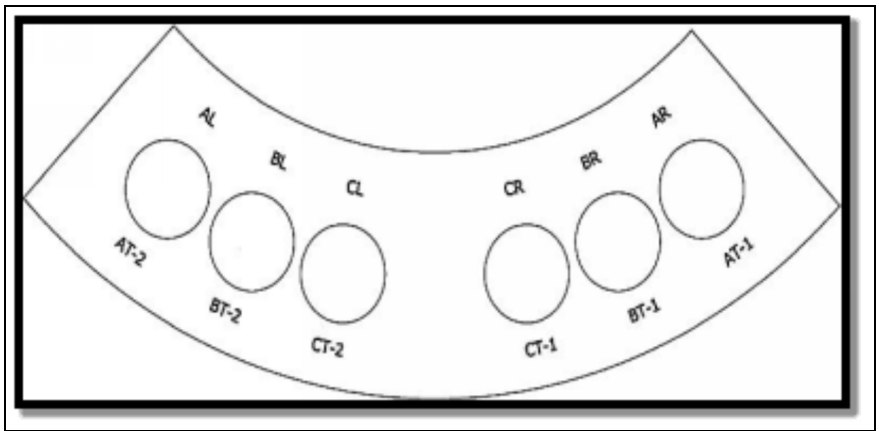
- Set the grid spacing to 1 unit and disable snap to the grid at a resolution of 0.125 units.

- Create three concentric circles with diameters of 5.0, 4.5, and 3.0. You can trim the bottom of the circle later.
- To accurately draw the arrow-shaped tick at the top of the 3.0 circle, it is recommended to zoom in for better precision. Next, create a vertical line with a length of 0.50 units directly below the previous line. Then, place the number 40 above it, ensuring that the text is centered.
- To achieve the desired arrangement, you can use angles of 135° and -135° to array these three objects around the perimeter of the 3.0 circle.
- To modify the arrayed numbers, use the **TEXTEDIT** command and set them to 0, 10, 20, 30, and so forth. You can perform all of these actions without exiting the command.
- Place the 0.25 vertical tick precisely above the 0.50 mark at the top center and create an array of it to the left and right. The dimensions should be 20 units in both directions.
- Position the needle in a horizontal orientation, ensuring it spans across the center of the dial.
- Create two duplicates of the dial and utilize the SCALE command to proportionally reduce their size, following the example provided. Next, carefully reposition the elements into their precise locations.
- Adjust the positions of the three needles to match the indicated orientation.




# Drawing: Stamping [INTERMEDIATE]

This drawing presents some challenges that may not be immediately obvious. There are multiple approaches to accomplish this task, utilizing various commands and techniques that you are already familiar with. It is important to note that to make the most of the suggestions, it will be necessary to adjust the limits.




**Here are some suggestions for creating drawings:**

Set the grid spacing to 0.50 units, the snap spacing to 0.25 units, and the limits of the drawing area to (**0,0**). The coordinates (**24,18**) represent a point in a two-dimensional space. Create two circles with radii of 10.25 and 6.50, positioned at approximate coordinates (**13,15**). These are later trimmed.

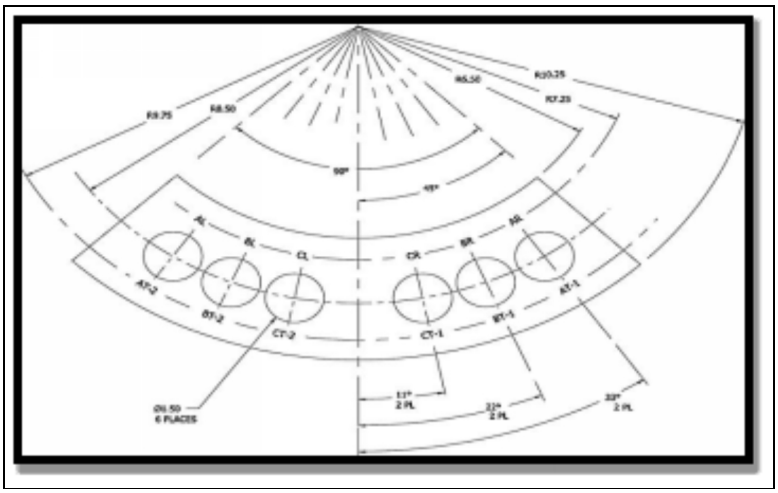
Create a vertical line extending downwards from the center point to the outer circle. Use the **COPY** and **ROTATE** commands to create the desired shape for the stamping ends. Use the **Copy mode** within the Rotate grip edit mode to effortlessly generate duplicates of the line, rotated at both 45° and -45° angles. The coordinate display indicates a value of 315°. Use the **TRIM command** to carefully remove any unnecessary lines and circles, leaving only the precise outline of the stamping. Create a circle with a diameter of 1.50 units at the center of the stamping, positioned 8.50 units below the coordinates (13,15). Create text that is aligned in the middle, with an absolute reference point at 7.25 units below the origin, and another absolute reference point at 1 unit below a point located at 9.75 units (13,15). To create offset copies of the circle and text, follow the procedure outlined in the next subsection. Afterward, use **TEXTEDIT** to make any necessary modifications to ensure that all text aligns with the drawing.

# Grip Copy Mode with Offset Snap Locations

This is a great chance to explore another grip editing feature. When you press the <Shift> key and select multiple copy points, AutoCAD will ensure that the copies are placed at equal distances from each other, based on the offset of your first two points.

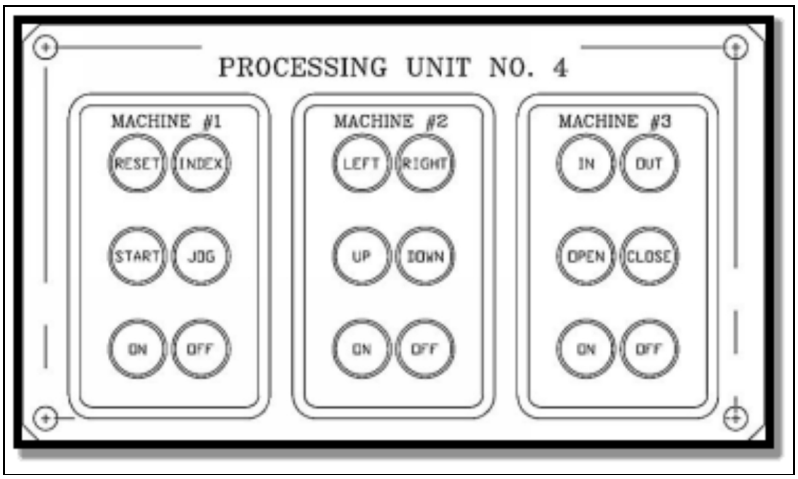
**Here's an example of the steps you can try:**

- Choose the circle and text.
- To begin grip editing, simply select any grip.
- Choose the **Rotate option** from the shortcut menu.
- Press **'b'** or choose the Base Point option from the shortcut menu.
- Specify the base point as the center of the stamping at coordinates (13,15).
- To copy, you can either press the "**c**" key or choose the "**Copy**" option from the shortcut menu.
- To rotate a copy 11° from the original, simply hold down the **Shift key** and move the cursor.
- Ensure that you maintain the <**Shift**> key pressed while maneuvering the cursor to generate extra duplicates. The copies are offset at an angle of 11 degrees.



# Drawing: Control Panel [INTERMEDIATE]

When executed accurately, this drawing provides a comprehensive demonstration of the commands at your disposal, showcasing their impressive capabilities. Make sure to utilize the **ARRAY** and **TEXTEDIT** commands in the manner explained. Additionally, it is advisable to consider the suggestion of adjusting the origin before commencing the task. Adjusting the origin in this drawing enhances the legibility of the dimensions, which are presented in ordinate form, measured from the (0,0) point situated at the lower-left corner of the object.




**Here are some suggestions for your drawings:**

- Set the grid spacing to 0.50 units and the snap spacing to 0.0625 units.

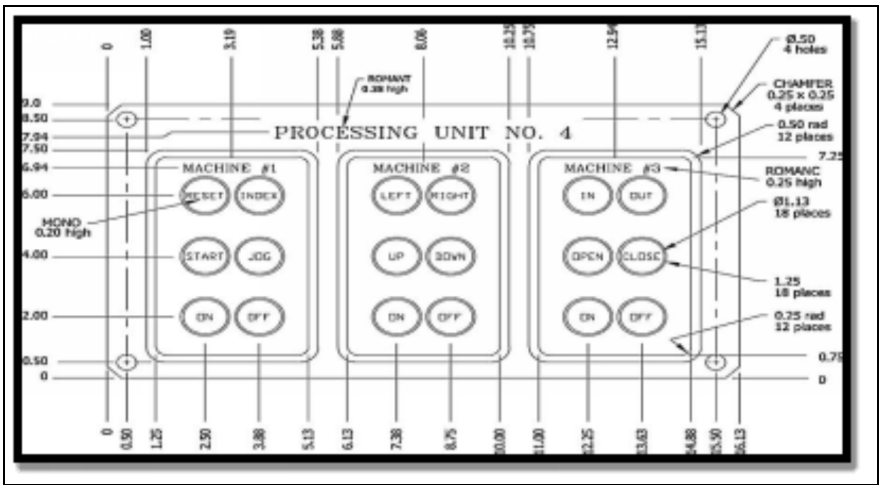
- Once you have completed the chamfered outer rectangle, proceed to draw the double outline of the left button box. Remember to fillet the corners for a polished finish. Take note of the varying fillet radii.
- Create the **On button** and position its text at the bottom left corner of the box. To achieve the desired layout, you can use the array command in AutoCAD to arrange the other buttons in the box in a 2 × 3 configuration.
- To modify the text on the lower-right button, use the **TEXTEDIT** command. Replace the existing text with "**Off**". Additionally, generate a text object at the top of the box containing the phrase "**MACHINE** #".
- Use the **ARRAY command** to create additional boxes by specifying a 1 × 3 array.
- Use **TEXTEDIT** to modify the text for additional buttons and machine numbers as demonstrated.

# Adjusting the Origin using the UCS Command

The drawing's dimensions are displayed in ordinate type, with measurements originating from a single point in the lower-left corner. This effectively establishes a new coordinate origin. By aligning the origin with this specific point, you will be able to easily retrieve dimension values directly from the coordinate display. To achieve this, you can simply set the lower-left limits to (-1, -1). Nevertheless, there is a more efficient way to accomplish this task by using the **UCS command**, which allows you to establish a user coordinate system with the origin at a point of your choosing. Extensive discussion on user coordinate systems is not provided in this section.

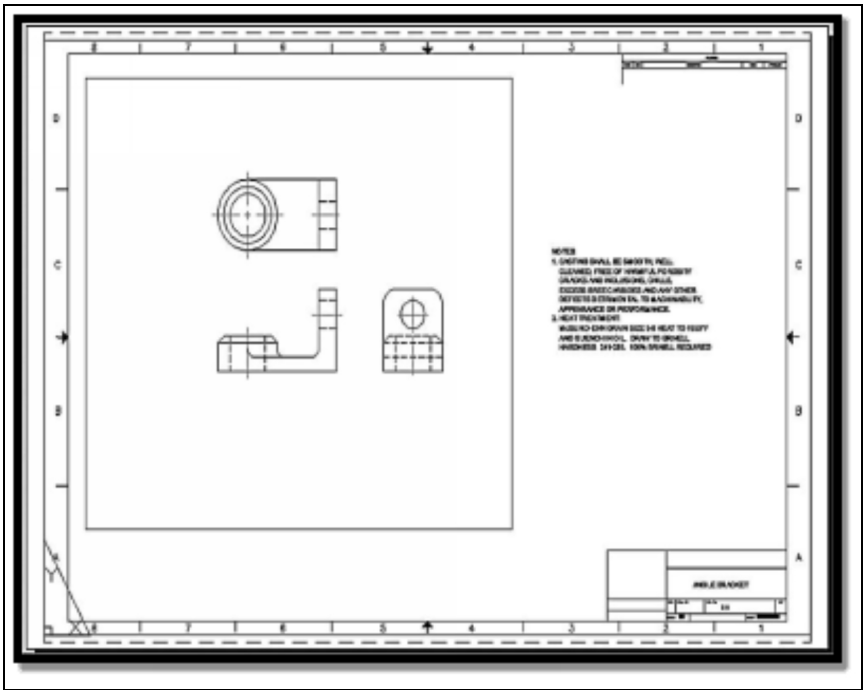
**Here is a straightforward procedure:**

Enter "**UCS**" at the command-line prompt. To specify the origin, either press the Enter key or choose the "**Origin**" option from the command line. Indicate the desired location for the new origin. That's the basic process. Position your cursor at the new origin and observe the coordinate display. The coordinates should be displayed as **0.00, 0.00, 0.00**, with all measurements taken from that point.




# Drawing: Angle Bracket [ADVANCED]

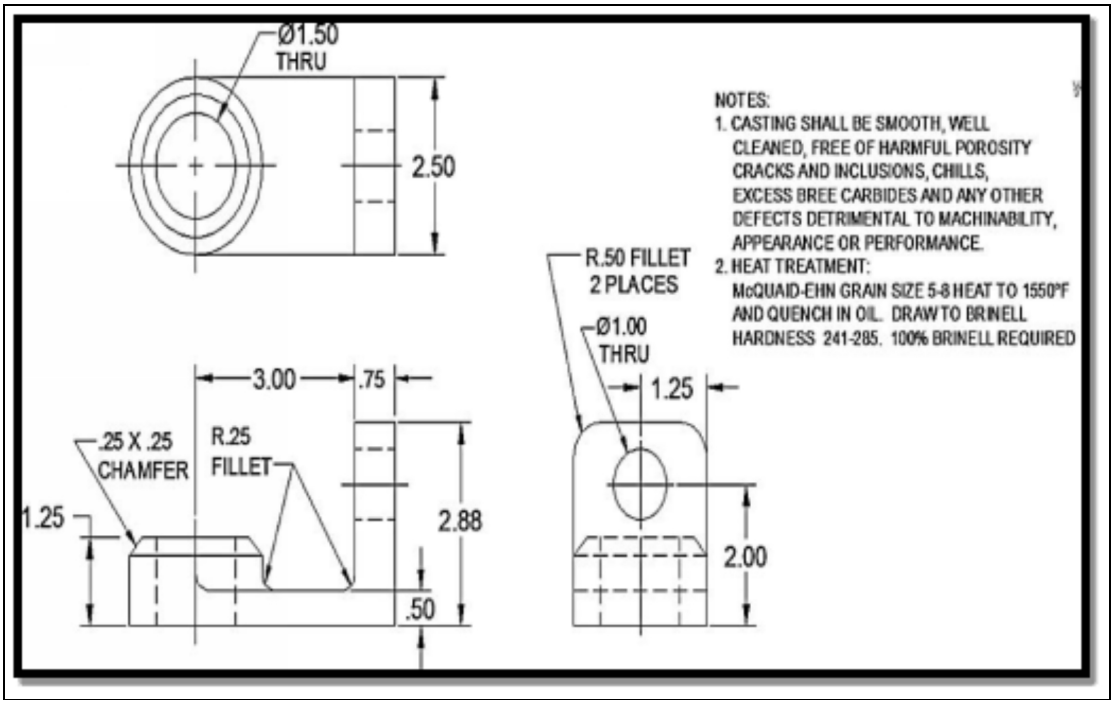
This drawing provides an opportunity to enhance your skills in working with multiline text (**mtext**) and using a drawing template that includes a title block and border. The drawing consists of three views and is generated in model space. However, it is displayed within a floating viewport on a paper space layout. When working with AutoCAD, it's important to remember that multiline text should be drawn in the layout view in paper space, rather than in model space.



**Here are some suggestions for your drawings:**

- Use the **Tutorial-iMfg drawing template** to create this drawing.
- When creating views in model space and placing text in paper space, it is necessary to switch between model space and the layout view.
- After opening the drawing with the D-Size Layout layout selected, the title block and border should be visible. Create a single floating viewport for your three views using **MVIEW**, just like in the reference drawing.
- To add the two notes in paper space on the layout, you can use the **MTEXT command**.

The dimensions provided in the drawing serve as a reference and should not be considered as part of the final design.




OceanofPDF.com

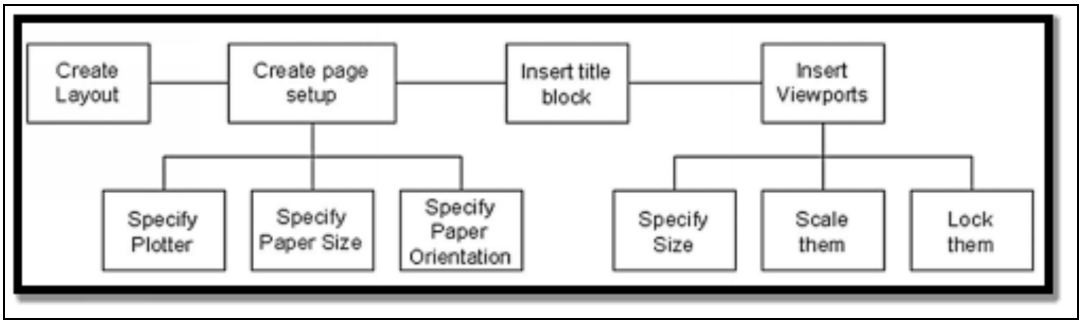
# CHAPTER TEN
# PLOTTING

## Introduction to Paper Space and Model Space

AutoCAD offers two distinct spaces for your drawings: Model Space and Paper Space. Model Space is where you create your drawing, while Paper Space is where you plot and prepare your drawing for printing. Each drawing contains a single Model space, while multiple Paper spaces, known as layouts, can be found within a file. Every layout is connected to a Page Setup, just like in other computer software. When it comes to plotting in AutoCAD, it's important to provide all the necessary details in the Page Setup. This includes specifying the plotter, paper size, and paper orientation (Portrait or Landscape). Once the layout is created and linked to the page setup, the next step is to insert a title block and then incorporate viewports, which are used to represent a specific section of the Model space. Next, the user will configure the scale of the viewports.

## What is Layouts?

The layout is the designated area where you can conveniently plot your drawing. Every layout will be connected to a Page Setup, Objects (such as a title block), text, dimensions, and **Viewports**, which will be discussed separately in the upcoming section. Take a look at the roadmap depicted below:




It is important to assign a name to each layout. When you start a new drawing using the acad.dwt template, you will automatically have two

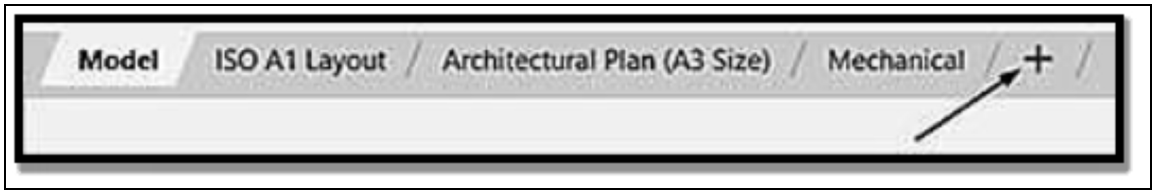
layouts: Layout1 and Layout2. **To utilize these two preset layouts, follow these steps:**

- Change the names to the preferred name.
- First, you'll want to establish the page setup and then connect the layout to it. Alternatively, you can easily link an already existing page setup.
- Place the title block into the drawing.
- Insert viewports, adjust their scale, and secure them in place.

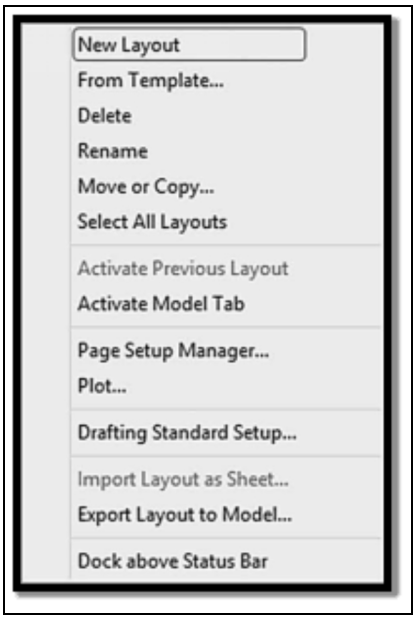
# Creating New Layout from the Beginning

This method will allow you to effortlessly generate a brand-new layout from the ground up. **There are several methods at your disposal:**

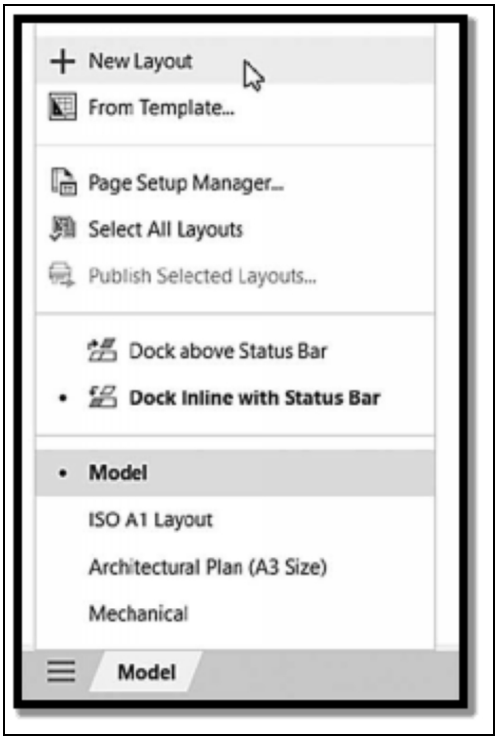
- To add a new layout, simply click on the **(+) sign** located to the right of the existing layout names. Take a look at the illustration provided:




- To create a new layout in AutoCAD, simply right-click on any existing layout name. This can be found as a tab at the lower left corner of the screen, just above the command window. A shortcut menu will appear, and from there, you can select the "**New Layout**" option.

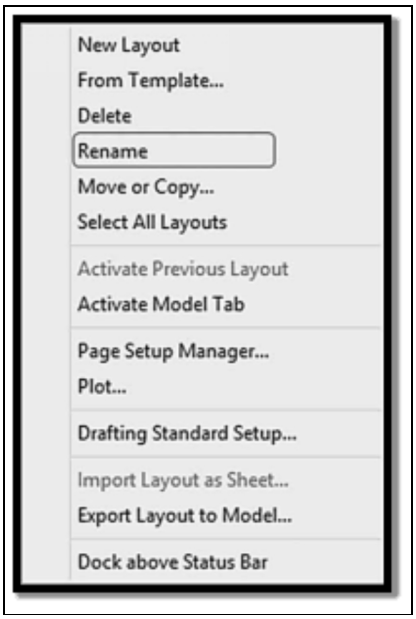



- To access the desired menu, navigate to the **Layout Tab Menu** located on the left side of the Model tab. From there, choose the **New Layout option**.

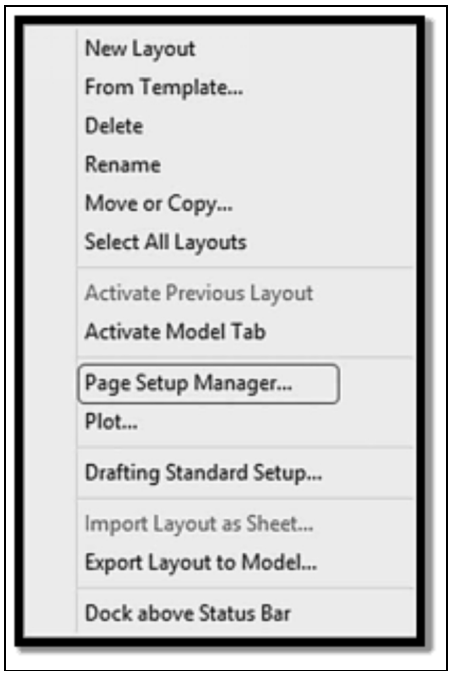



**Once the new layout has been created, proceed with the following steps:**

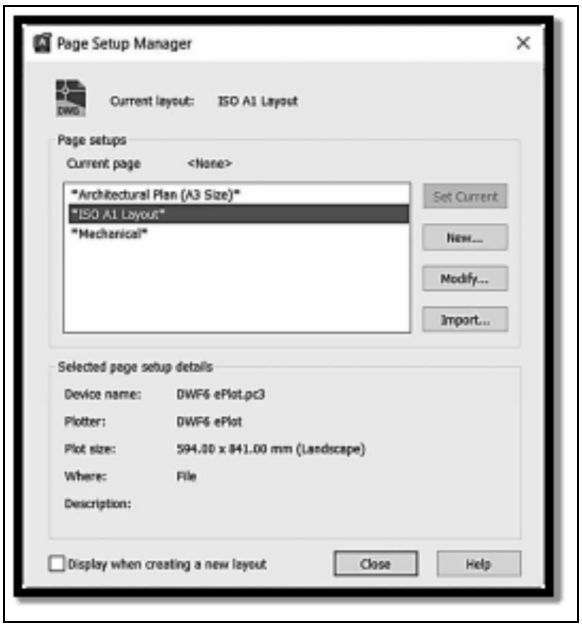
- A new layout will be created and assigned a temporary name. To rename it, simply right-click on the item and choose the **Rename option**. Alternatively, you can double-click on the temporary name to make it editable and enter the new name.



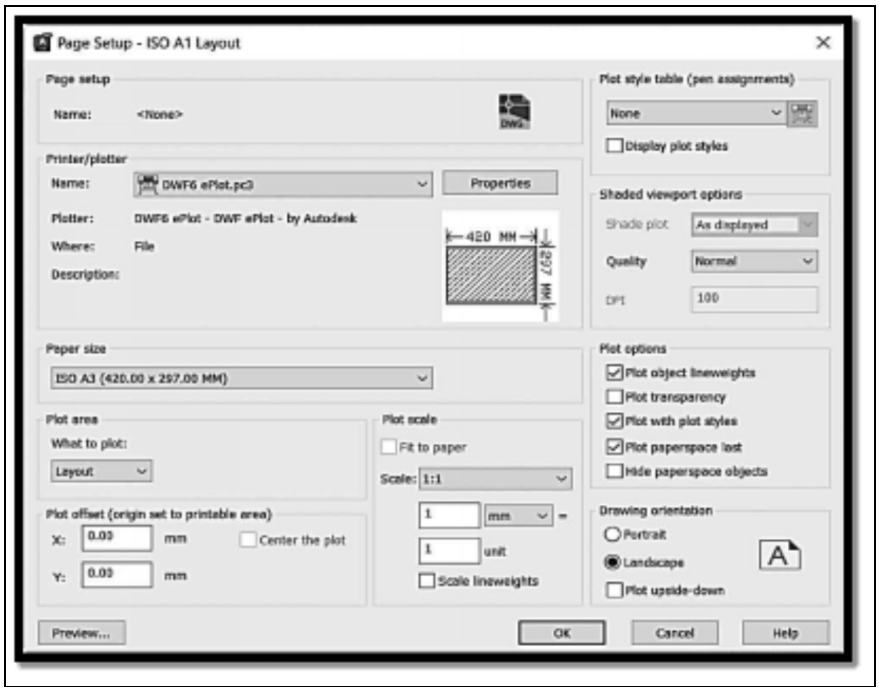

- Connect the new layout to a page setup. To accomplish this, simply perform right-click on the layout name and proceed to choose the **Page Setup Manager**.

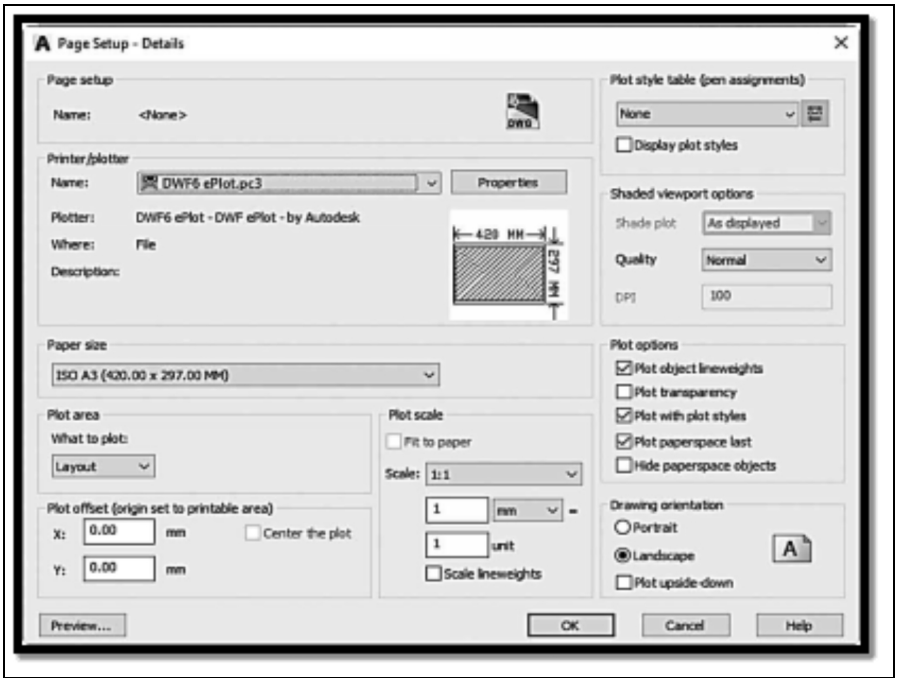


**A dialog box will appear:**




In the dialog box, you'll find the Current layout name at the top, while at the bottom, there's a summary of the current page setup called Selected page setup details. At last, you'll notice a checkbox labeled "**Display when creating a new layout**". Enabling this checkbox will ensure that this dialog box pops up every time you create a new layout. By default, AutoCAD will automatically generate a page setup with the same name as the layout. To make changes, simply click on the **Modify button**. This action will prompt a dialog box to appear, displaying the options available for modification.




- Choose the plotter you want to use, ensuring that it is properly connected and configured.
- Choose the **paper size** that you want.
- When selecting what to plot, it is important to always choose "**Layout**" as the option. This is because the other choices are specifically for printing in Model space.

- When printing from the layout in AutoCAD, it is recommended to keep the **plot offset** values as zeros.
- When it comes to printing from layout, it's important to consider the input plot scale. In this case, viewports play a crucial role as each viewport will have its scale, which we will discuss in detail in our next topic. Therefore, it is necessary to adjust the layout plot scale to 1=1. Also, please indicate whether you would like to include **line weights** when scaling.
- **Plot-style tables,** also known as pen assignments, will be covered in detail towards the end of this chapter.
- **Shaded viewport options** are specifically designed for 3D modeling in AutoCAD.
- Ensure that the **plot options** are set to their default values.
- Choose the drawing orientation, either portrait or landscape.
- When printing with the Plotter, you have the option to select whether it prints from top to bottom. Simply check the checkbox if you want it to do so.
- Clicking OK will save the Page Setup you have created, making it accessible for all layouts in the current drawing file.
- After completing the previous steps, you will be directed back to the initial dialog box. From there, locate the page setup option and choose "**Set Current**" by either clicking on it or double-clicking on the name of the Page Setup. The current layout is now associated with the page setup you choose.
- To modify the settings of an existing page setup, you can simply click on the "**Modify**" option. To import a saved Page Setup from an existing file, simply click on the **Import** option.

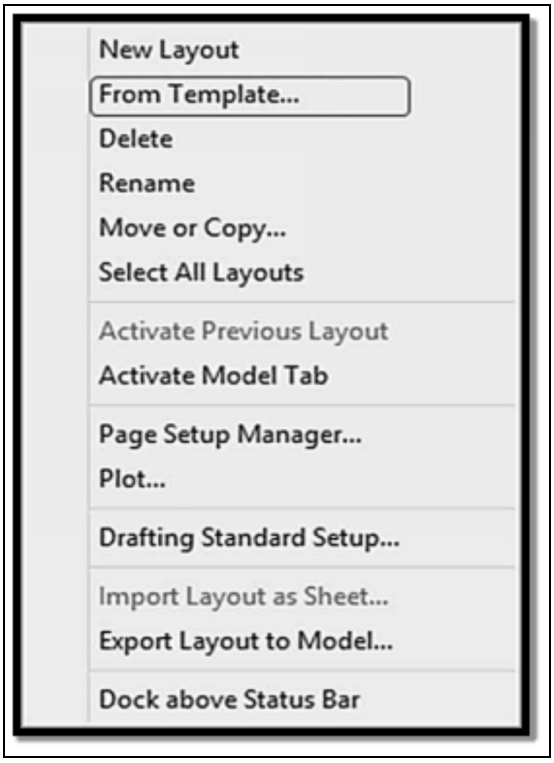
# Creating a New Layout Using a Template

This procedure allows for the seamless importation of layouts from a template file, encompassing all aspects such as page setup, title block, viewports, text, and more.

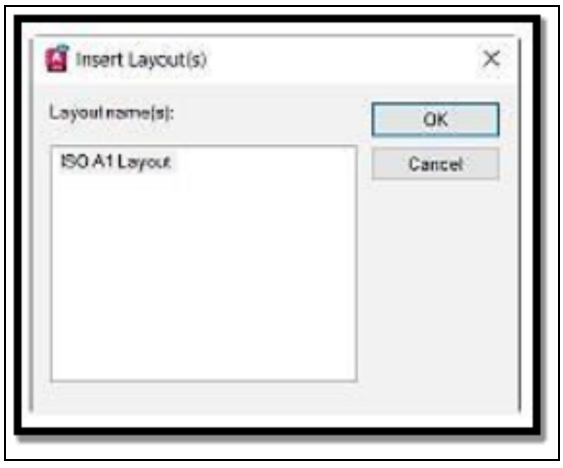
**To accomplish this, you will need to follow these steps:**

- To access the menu, simply perform right-click on any existing layout. From there, you will be presented with the option to

select "**From template**".




- The standard Open file dialog box will appear, allowing you to choose the template file you want. Choose the template you want and click on **Open**. Once you do that, a dialog box will appear:




- Choose one of the layouts from the list and click OK. Upon importing the layout, it will become apparent that it now exists within your drawing.

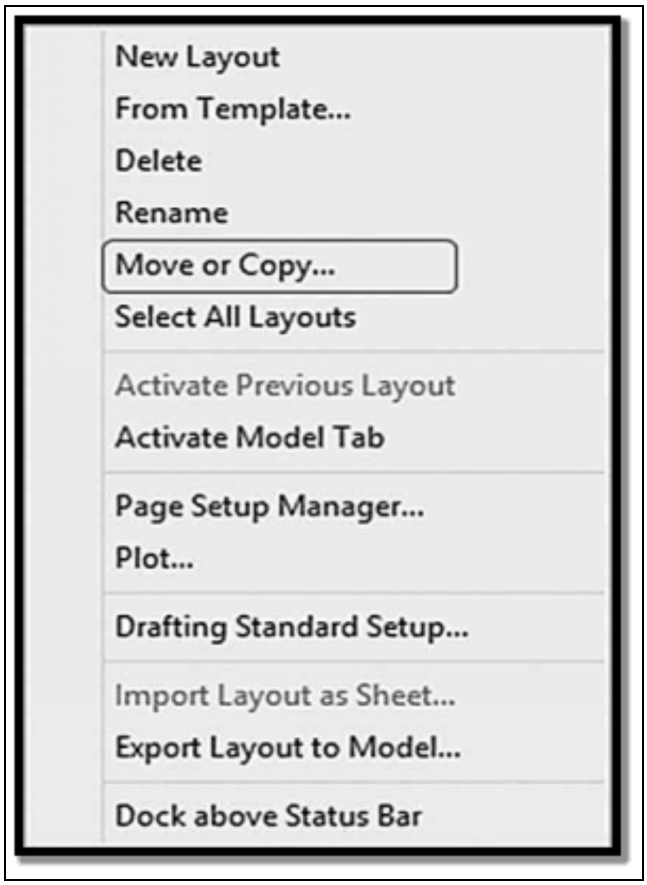
# How to Create Layouts Using Copying

There are two methods available to duplicate an existing layout. One approach is reminiscent of a technique commonly used in Microsoft Excel.
**Here are the necessary steps:**

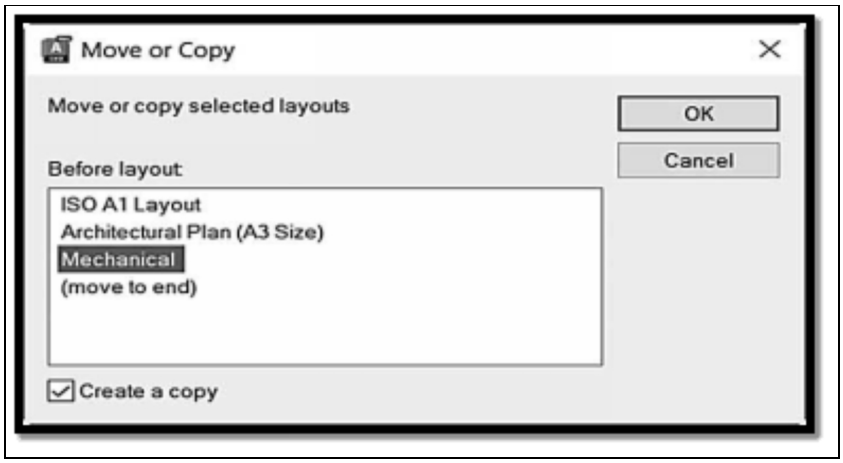
- Choose the layout you want.
- Hold down the **Ctrl key** on your keyboard and simultaneously drag the mouse to move the copied layout to its new position.

- Modify the name of the new layout

To copy layouts, simply select the layout you want and then right-click. From the shortcut menu, you can choose the **Move or Copy option**:



**A dialog box will be displayed for your reference:**




Displayed is a list of the current layouts. Simply select one of them and proceed by clicking on the checkbox labeled "**Create a copy**." You have the option to create a duplicate, which can be renamed and modified as needed. With the same option, you can effortlessly shift a layout either to the left or to the right from its current position. Another way to move a layout is by manually clicking and dragging the layout name to the desired location.

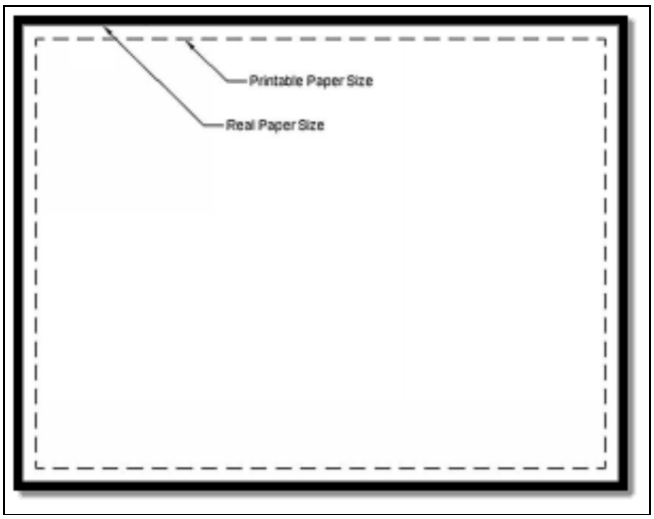
**Take note of the following:**

To rearrange the position of the layouts, you can easily move them by holding the name in the tab and then clicking and dragging it.

**At this point, you have completed the following steps:**

- Create a new layout.
- Creating a Page Setup in AutoCAD.
- Linking a Page Setup to the layout.

**Once you choose the newly created layout, you'll be able to observe the following:**




The outer frame represents the actual paper size, while the inside frame (indicated by the dashed line) represents the printable paper size. The printable paper size is calculated by subtracting the printer's margins from the paper size. With this layout, you can easily determine which objects will be printed and which ones won't. Anything outside the dashed line will not appear in the final print. This demonstrates that when printing from layouts, the output will accurately reflect what is displayed on the screen. To access the mentioned commands, navigate to the **Layout tab**. Note that this tab will only be visible when you are on one of the layouts. Find the **Layout panel and observe the buttons displayed:**

Within this panel, you can create a new layout or use a pre-existing template. To access the page setup dialog box, simply click on the second button.

# How to Create Viewports

After opening a newly created layout, you will notice a solitary viewport positioned in the center of the paper size. A viewport is a window that displays the view of your Model Space. It is initially scaled to fit the size of the window. Viewports inserted in layouts can be tiled, scaled, and printed. **There are various methods available for adding viewports to layouts in AutoCAD:**

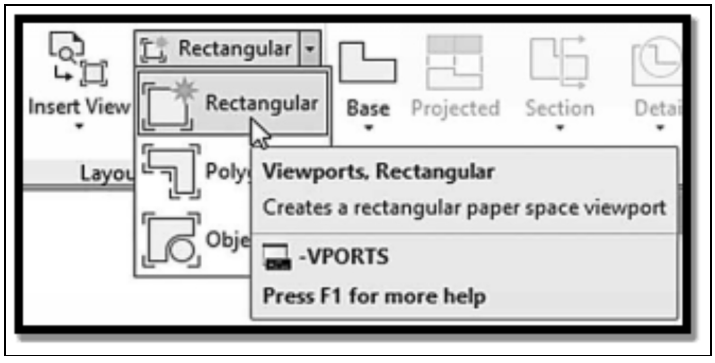
- Add a single rectangular viewport.

- Adding multiple rectangular viewports
- Clipping an existing viewport
- Adding a single polygonal viewport
- Converting the object to be a viewport

**Take note of the following:**
The Layout tab is only visible when you are working within a layout.

# Adding Single Rectangular Viewports

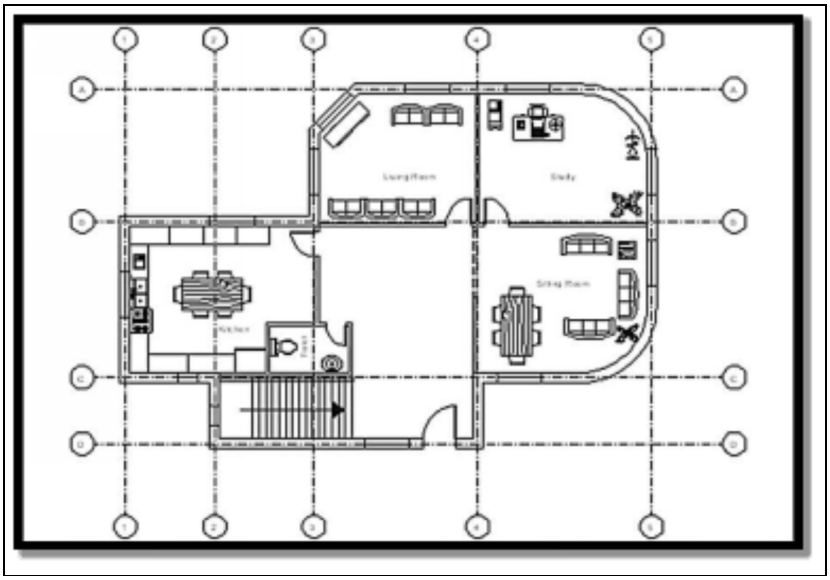
This command is used to insert individual rectangular viewports into a layout. When specifying the area of the viewport, it is important to specify two opposite corners. To execute this command, navigate to the Layout tab and find the Layout Viewports panel. From there, simply click on the **Rectangular button**.




**Here are the prompts that will be displayed:** Specify the corner of the viewport or choose from the available options such as ON, OFF, Fit, Shadeplot, Lock, Object, Polygonal, Restore Layer, 2, 3, or 4. If you prefer, you can simply select Fit. Once you've done that, please specify the

opposite corner: Indicate the coordinates of the two diagonally opposite corners to generate a rectangular viewport.

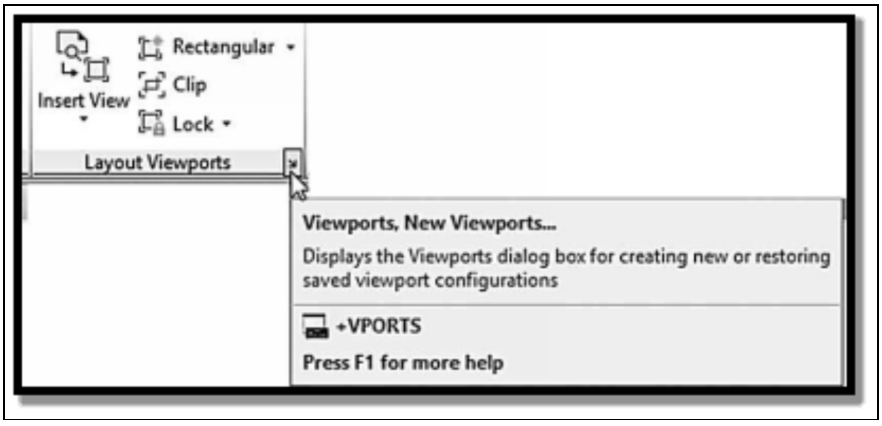
**Here is what you will receive:**



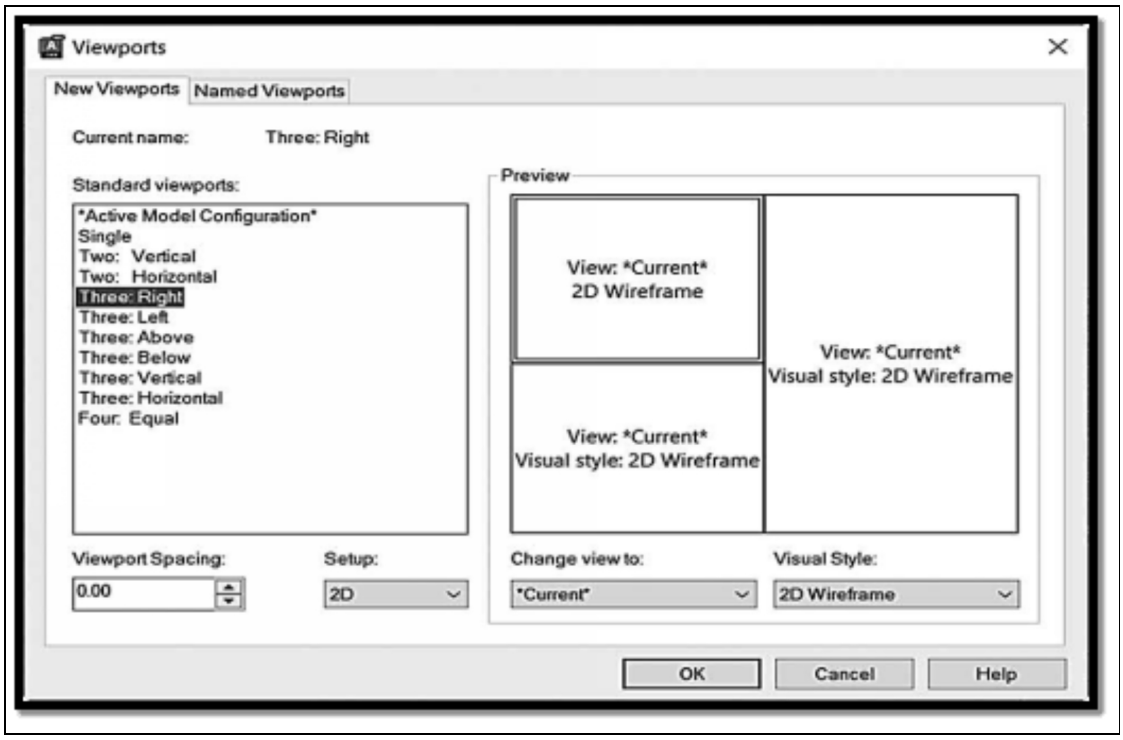
# How to Add Multiple Rectangular Viewports

To add multiple rectangular viewports in a layout, you can use this command. When specifying the area of the viewports, the user needs to specify two opposite corners. To execute this command, navigate to the **Layout tab** and find the Layout Viewports panel.

**From there, simply click on New Viewports, as illustrated in the image below:**

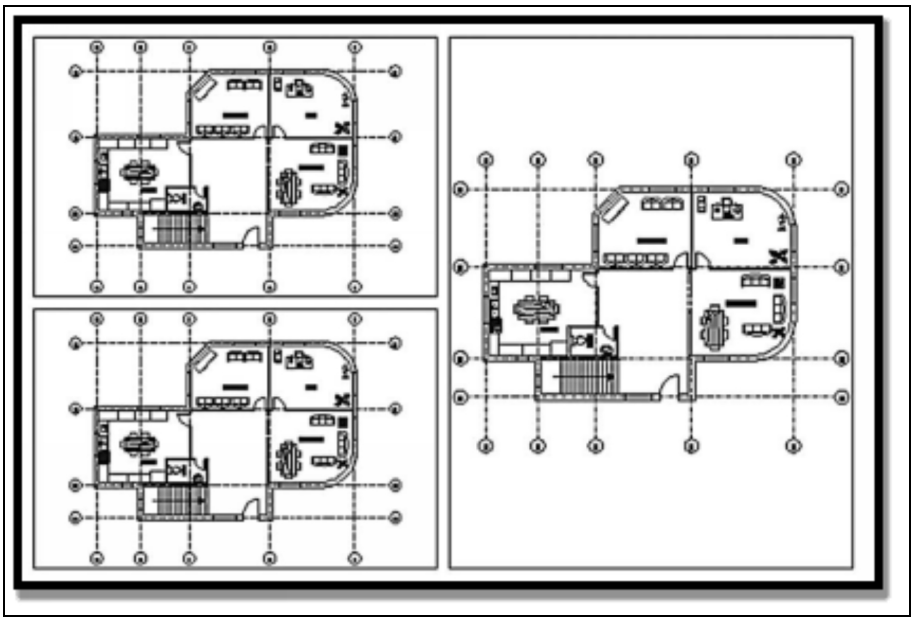



**A dialog box will be displayed for your reference:**



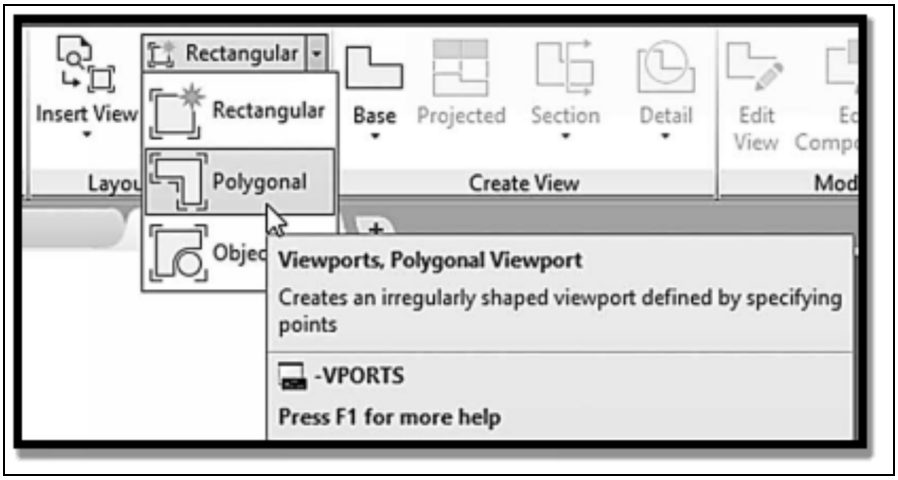

- Choose the preferred display option from the provided list. By default, the Viewport Spacing value is set to 0, indicating that the viewports will be tiled. If you wish to have the items divided, enter a value that is higher than zero. After clicking **OK**, a prompt will appear:
- Specify the coordinates for the first corner or select the "**Fit**" option:
- Specify the corner on the opposite side:

**Take a look at the illustration below:**



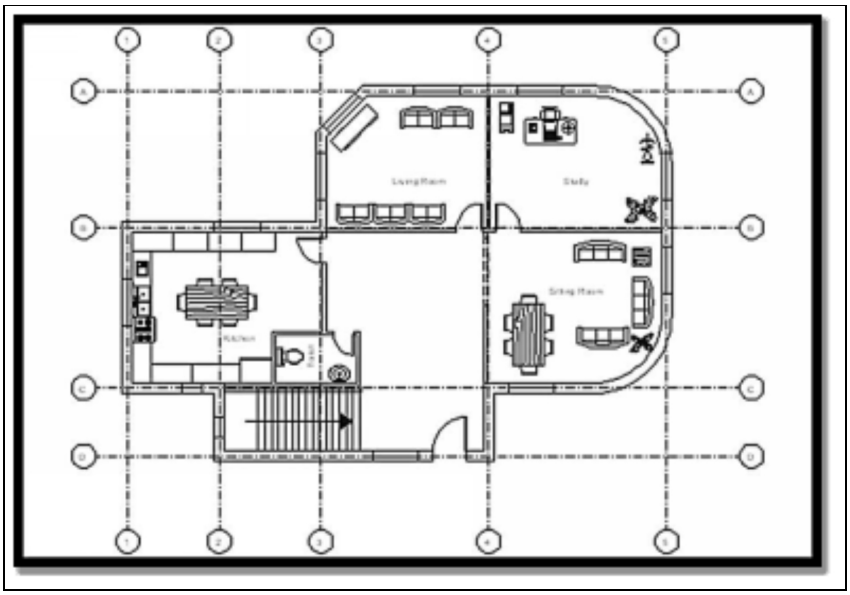
# Adding polygonal viewports

This command will create a polygonal viewport that includes both straight lines and arcs. To initiate this command, navigate to the **Layout tab** and locate the **Layout Viewports panel**. From there, simply click on the **Polygonal button**:



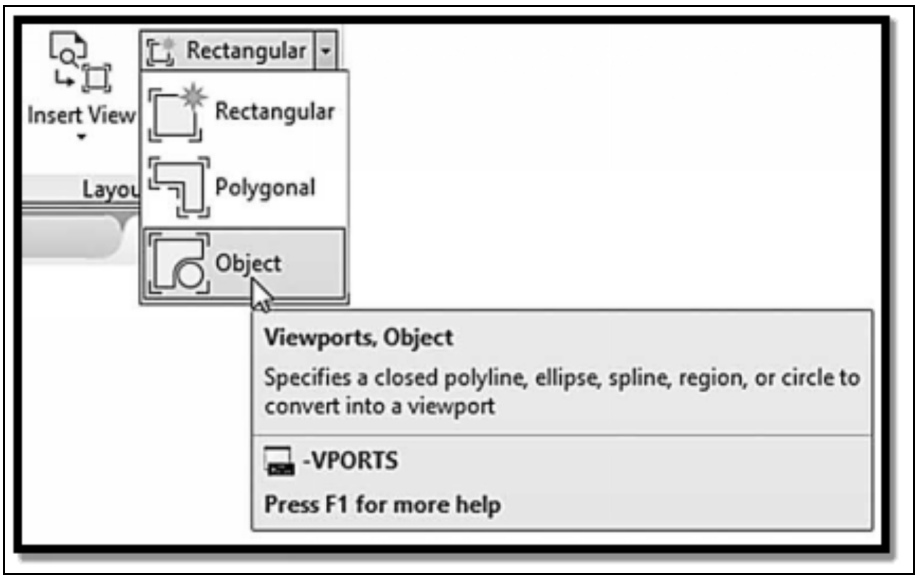

- **After launching the program, a series of prompts will be displayed:**
- ***Specify starting point:***
- ***Specify the next point or choose from the options of Arc, Length, or Undo:***

- ***Specify the coordinates for the next point or choose from the options: Arc, Close, Length, or Undo.***
- These prompts resemble the ones used in the Polyline command.
- **Observe the image displayed below:**

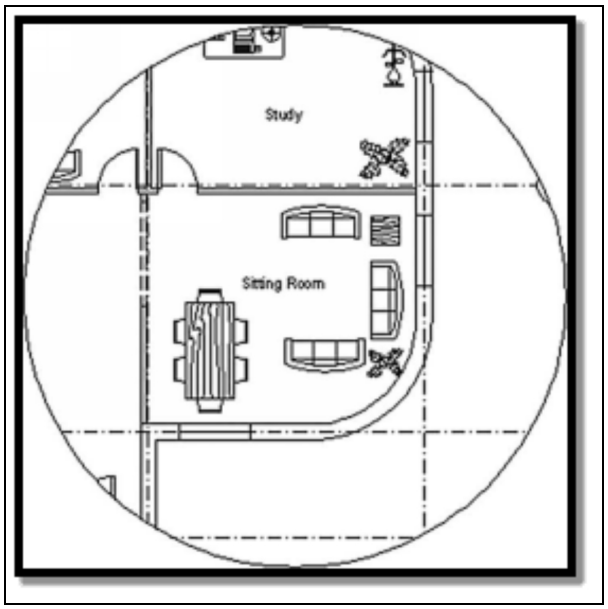


# Creating Viewports by Converting Existing Objects

To convert an existing object to a viewport, ensure that the object is a single object such as a polyline or circle (lines and arcs are not permitted). To initiate this command, navigate to the **Layout tab** and find the **Layout Viewports panel**. Next, click on the **From Object button**:



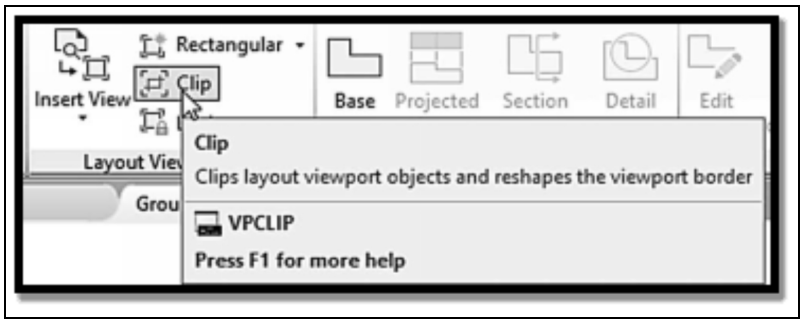

- A prompt will be displayed on your screen.
- Please choose the object you wish to use for clipping the viewport:
- **Observe the illustration provided:**



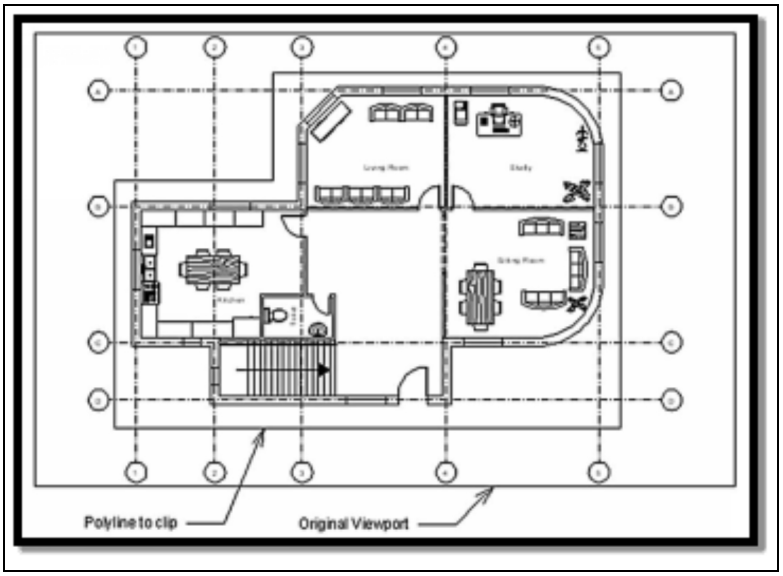

# Creating Viewports by Clipping Existing Viewports

The command will clip an existing viewport, resulting in the creation of a new shape. To initiate this command, navigate to the **Layout tab** and find

the **Layout Viewports panel**. From there, simply click on the **Clip button**.



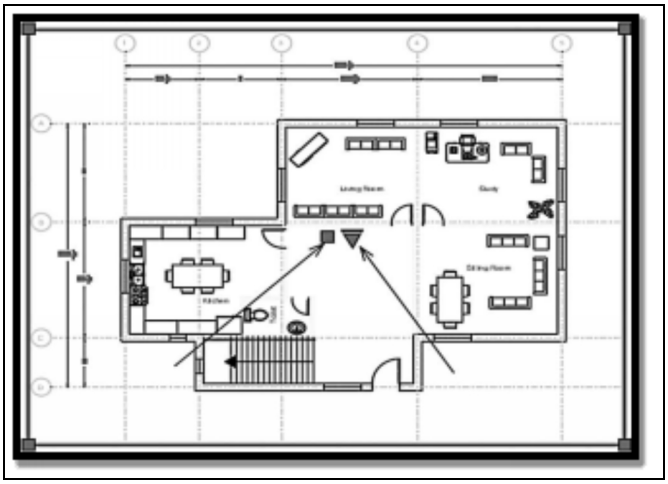

- A prompt will appear on your screen.
- **Choose the viewport you want to clip:**
- **Select the clipping object or choose [Polygonal] for a polygonal selection. Specify the start point and then provide the next point.**
- **Specify the next point or use the Undo command to go back.**
- **[Arc/Close/Length/Undo]:**
- To begin, choose a viewport that already exists. To select a previously drawn polyline or create an irregular shape, you have the option to use the Polygonal tool. This tool is similar to the **Polygonal viewport command** mentioned earlier. Observe the illustration provided:



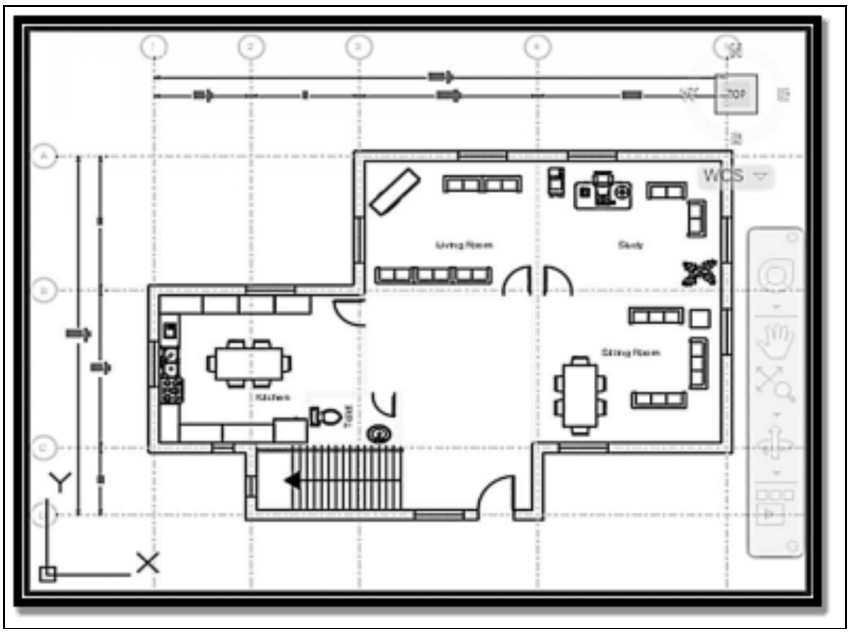
# Managing Viewports After Creation

**There are two different modes available for working with viewports:**

- When you're outside the viewport, you'll handle it just like any other object in your drawing. To perform various actions on the viewport, simply select it from its frame and use commands such as erase, copy, move, scale, stretch, rotate, and more.



- When working in the second mode, you can enter the viewport by simply double-clicking inside it. In this mode, you can manipulate the objects within the viewport by zooming, panning, scaling, and more. To switch back to the first mode, simply double-click outside the viewport.

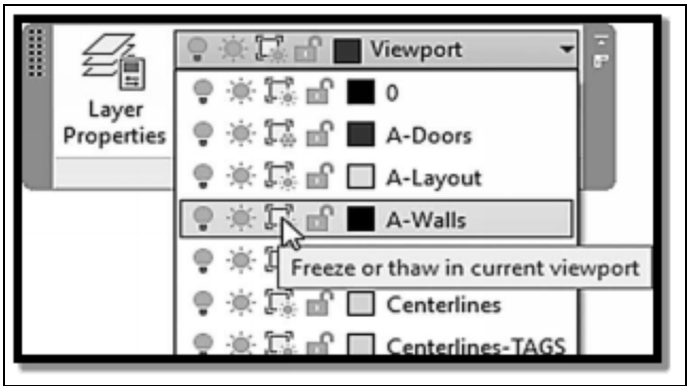


# How to Freeze Layers in Viewports

The Freezing function is capable of freezing layers in Model space as well as all viewports in all layouts.

**To freeze a layer in a specific viewport, follow these steps:**

- Double-click within the desired viewport to proceed.
- To freeze or thaw a layer in the current viewport, navigate to the **Home tab** and find the Layers panel. From there, select the layer list and click on the appropriate icon. Observe the illustration provided:




In addition, you can freeze or thaw layers in all viewports except for the current one. To accomplish this, open the Layer Properties Manager and

choose the specific layers you want. Right-click on the selected layers and choose the option to freeze them in the Viewports. Make sure to select the "**In All Viewports Except Current**" option for the desired effect.

# How to Scale and Maximize Viewports

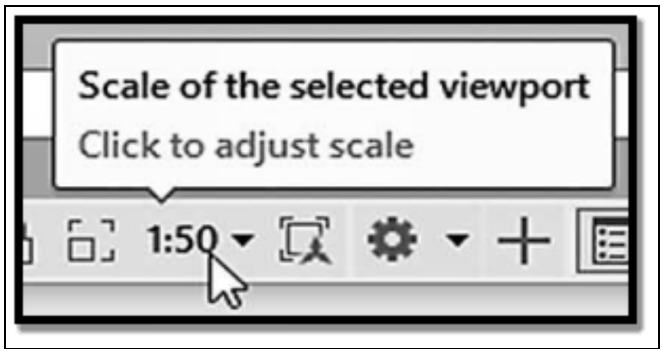
By default, when inserting a viewport, AutoCAD automatically adjusts the zoom level to fit the specified viewport size, ensuring the entire drawing is visible. It is important to note that the current viewport is not set to scale. To ensure accurate measurements, the user should manually set the scale relative to the Model Space units. **There are several approaches to accomplish that:**

# First approach

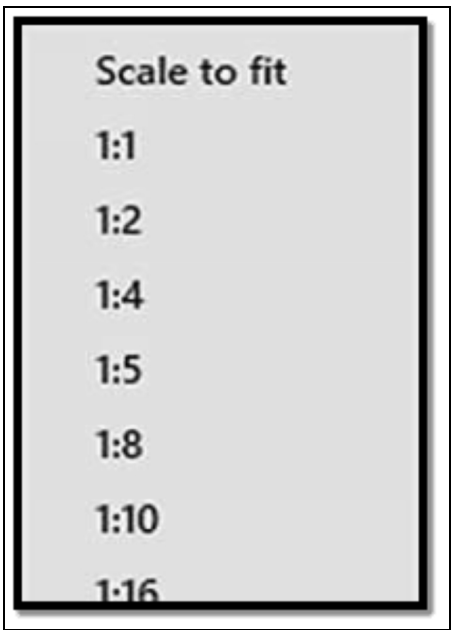
- Click on the viewport frame.
- The triangle will be displayed in the center of the viewport.
- Click on the triangle. You will see a list of scales. Choose the scale that you want
- The size of the viewport will adjust accordingly based on the scale.

# Second approach

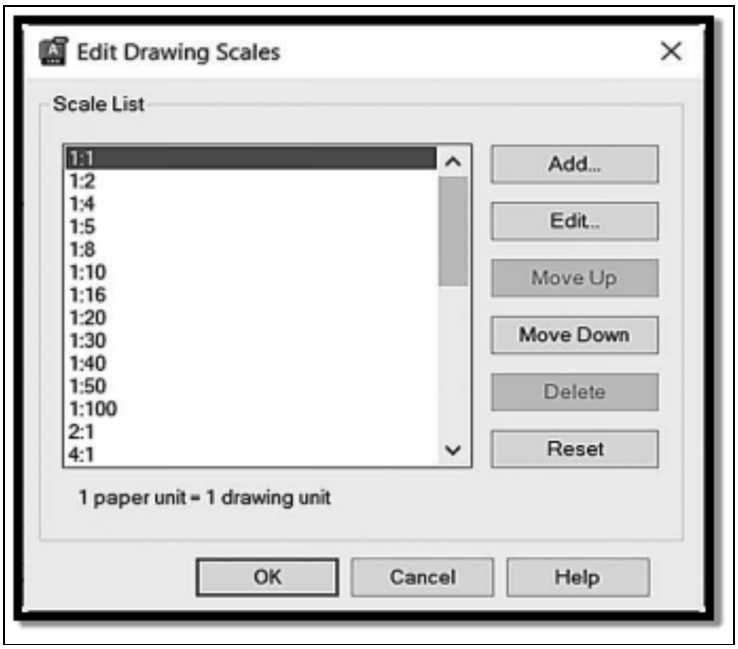
- Click twice within the desired viewport or select only the frame of the viewport.
- On the right side of the Status bar, you can find the **Viewport Scale list**.



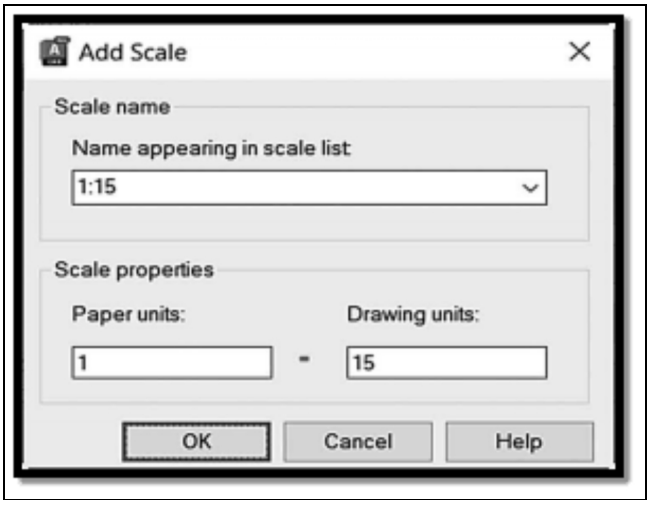

- **Select the list that includes all scales, like the one shown below:**




Choose the appropriate scale for your viewport. If the desired scale is not found, you can select the **Custom option**. This will bring up a dialog box with the following options:

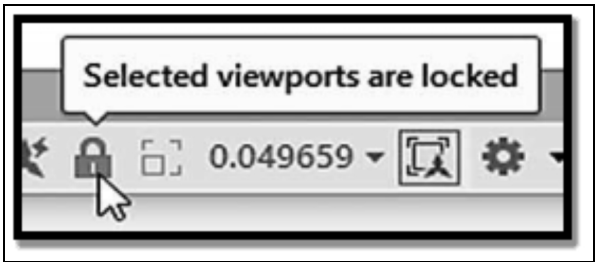



To add a new scale, simply click on the **Add button**. This action will prompt a dialog box to appear on your screen.

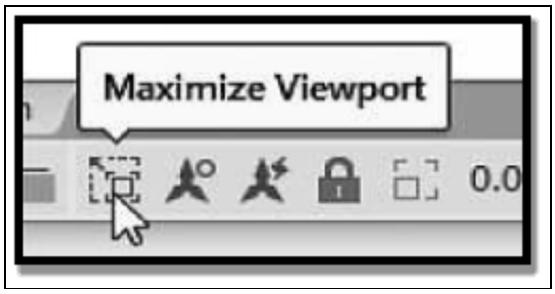



Enter the scale you want and then click **OK twice**. When working within the viewport, it is important to remember that after setting the scale, it is recommended to utilize the Pan command instead of the Zoom command. Using the Zoom command can disrupt the scale that you have already set. To resolve this issue, you can prevent any changes to the viewport display by selecting the viewport and clicking on the lock icon in the status bar. Make sure you are inside the viewport or have the viewport border selected.

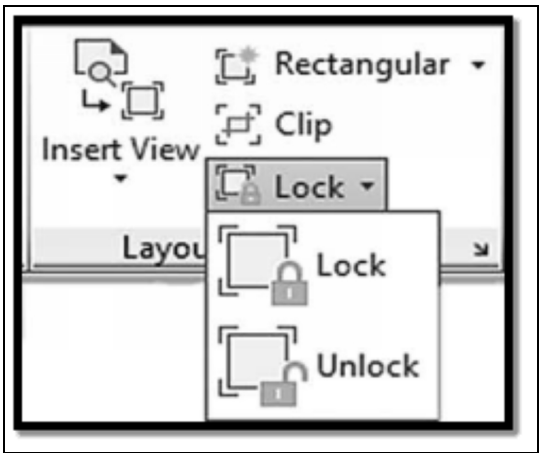
Once locked, the viewport will be protected and no modifications can be made.




The Maximize function allows you to temporarily adjust the viewport to perfectly fit the size of the screen. With this method, you can conveniently edit objects to your liking without having to switch from the layout to the Model space. Access the status bar and locate the **Maximize Viewport button**. Click on it to proceed:




Clicking on the identical button will effortlessly revert the viewport to its original size. Alternatively, the user can navigate to the **Layout tab** and access the **Layout Viewports panel**. From there, you can use a pair of buttons to perform the desired actions:



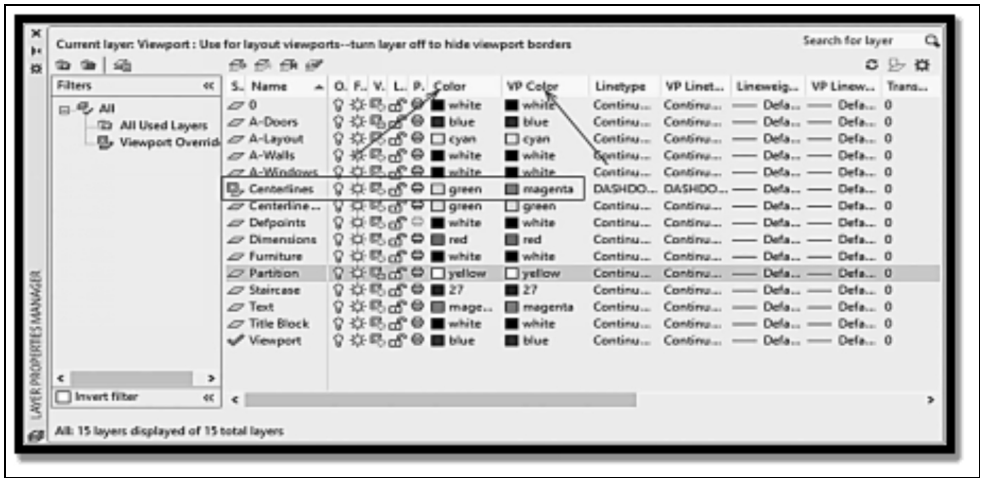
# How to Laver Override in Viewports

Consistency is key when it comes to displaying a layer in all viewports. Make sure to use the same **color, line type, line weight,** and **plot style**. To modify the display of this viewport, you can adjust these settings. This feature is commonly referred to as layer override.

**Follow these steps:**

- Double-click within the desired viewport to proceed.
- Open the **Layer Properties Manager command**.
- Make the desired changes under the options for VP Color, VP Linetype, VP Lineweight, or VP Plot Style.
- These modifications will only affect the current viewport.

**Take a look at the illustration below:**

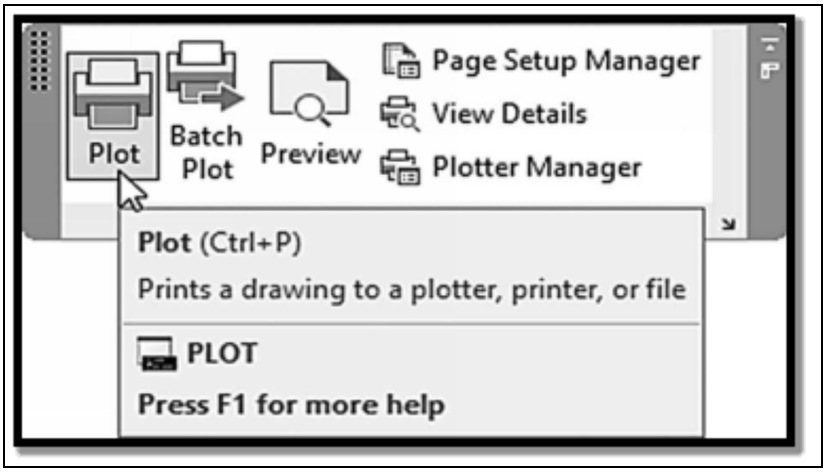



- In the given example, the layer named Centerlines has a color Green, which applies to both the Model Space and all other viewports. Additionally, in the current viewport, there is an override color of **Magenta**.
- Additionally, it is important to take note of the symbol located to the left of the layer name.

# How to Use Plot Command

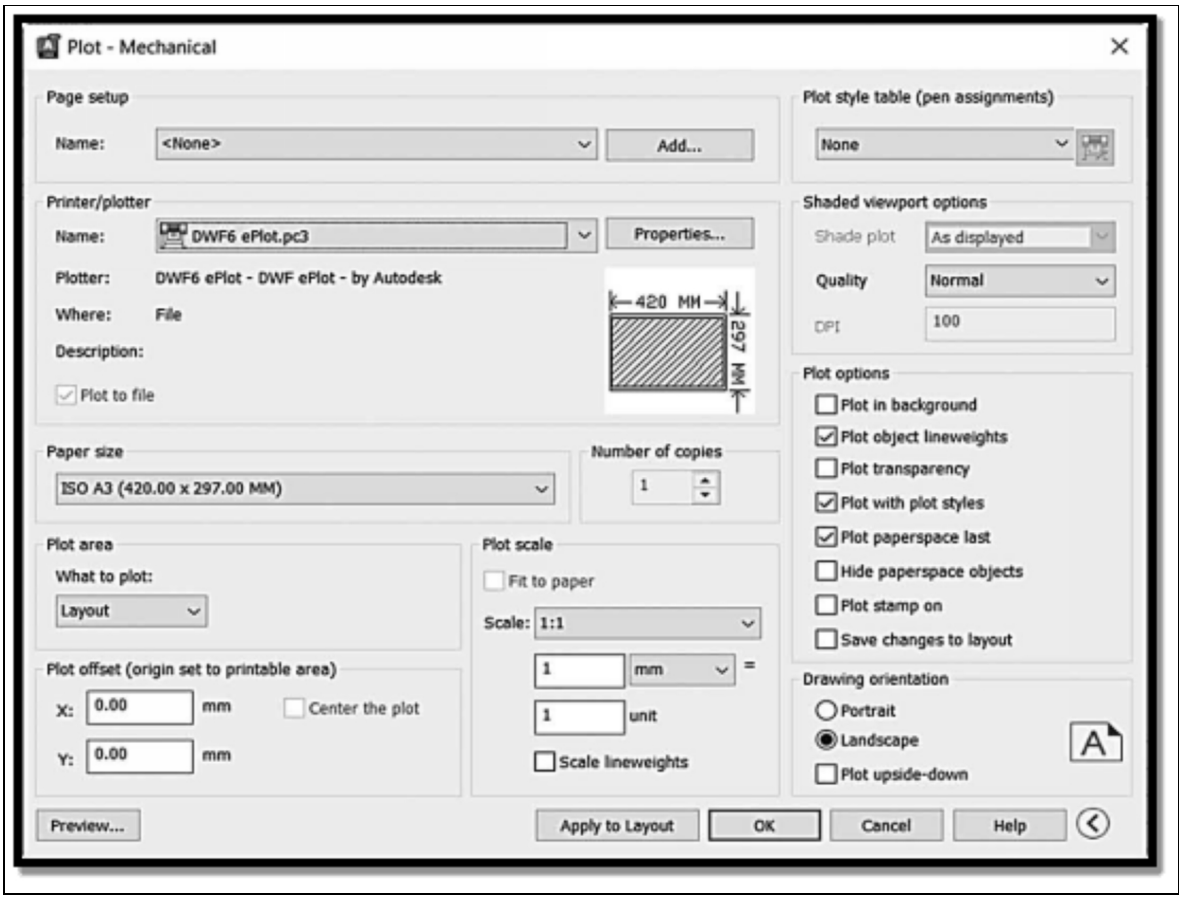
Now, let's proceed to the last step. The purpose of this command is to send the content set in the layout to the designated plotter. To execute this

command, navigate to the Output tab, find the **Plot panel**, and click on the **Plot icon**:




When you have multiple layouts in your drawing and you select the **Plot command** instead of the **Batch Plot command**, AutoCAD will display the following message:

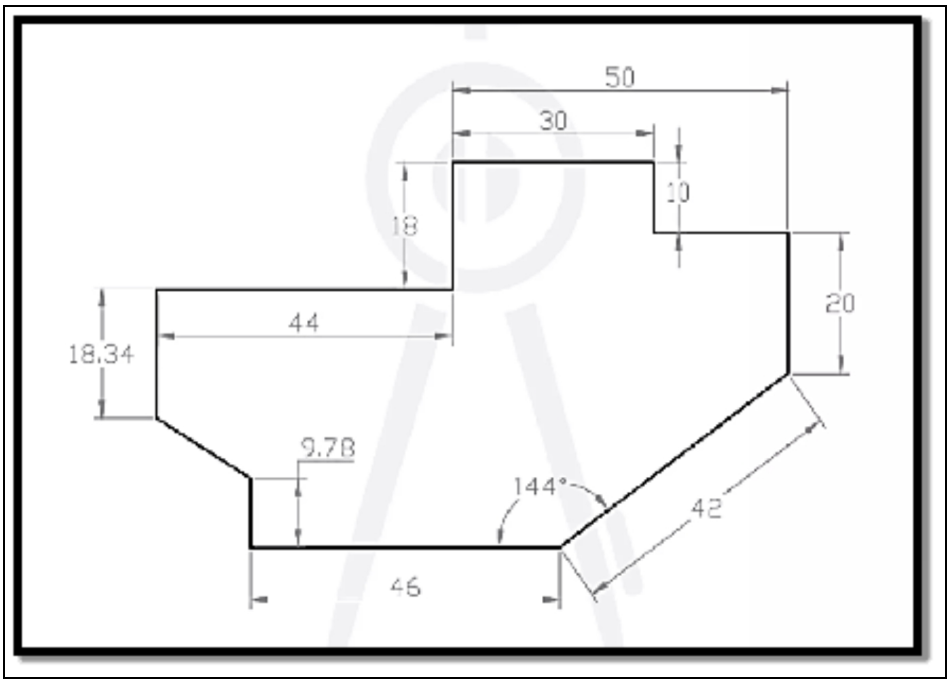
Explore the Batch Plot command to expand your knowledge, give it a try, or proceed with plotting a single sheet. When you decide to proceed with printing a single sheet, a dialog box will appear:

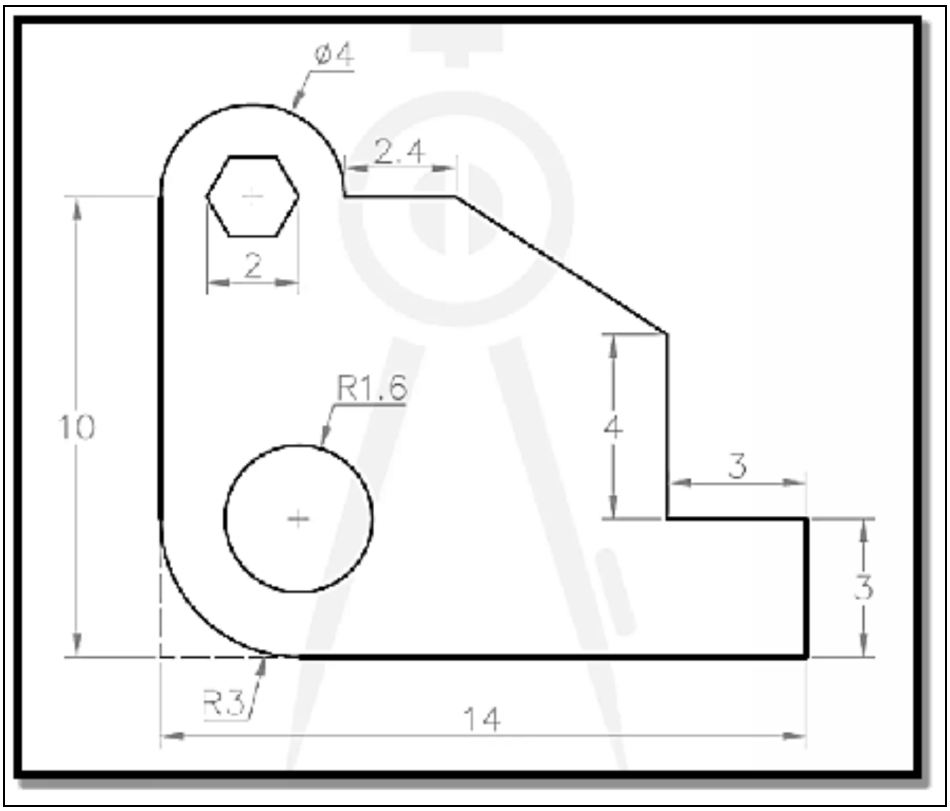


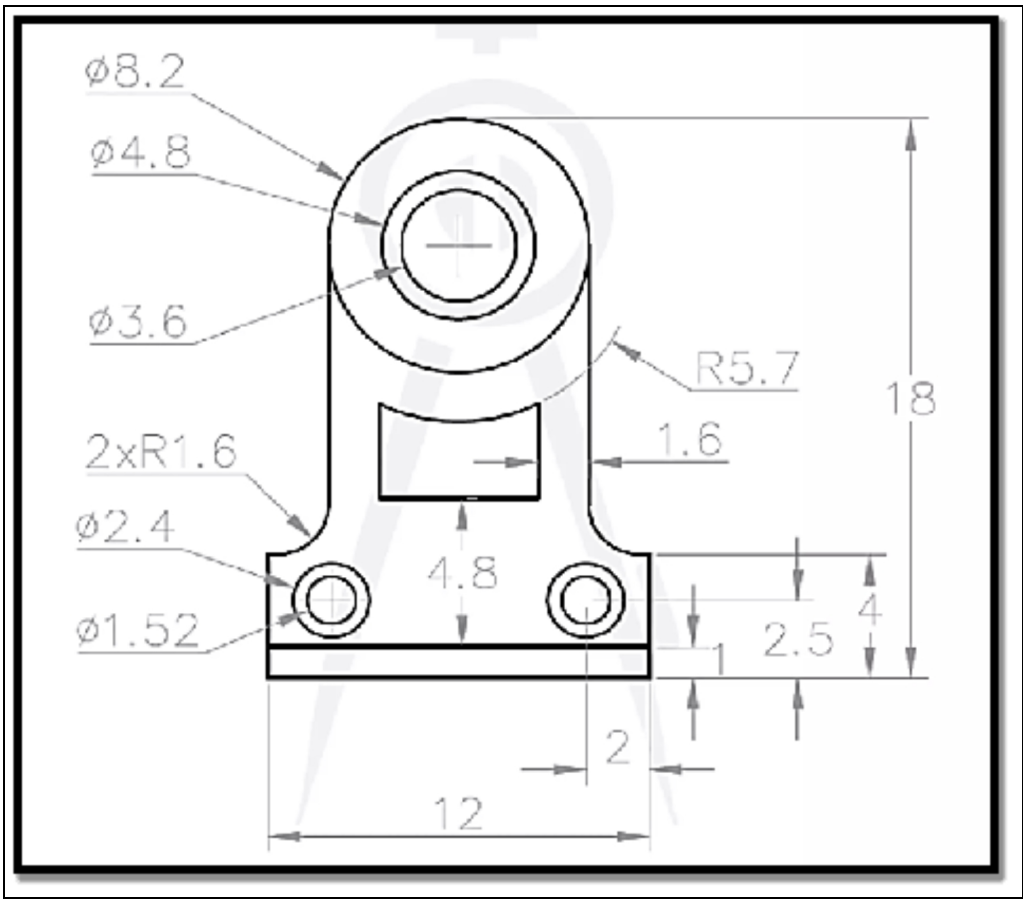

This dialog box is the same as the Page setup settings. Modifying any of these settings will cause AutoCAD to separate the Page setup from the current layout. To send the layout to the plotter, simply click the **OK button**. To preserve the configuration of this dialog box along with the layout, simply click on the **Apply to Layout button.** To view the final printed drawing on the screen before printing it out, simply click on the **Preview button**. To preview your drawing outside of this dialog box, navigate to the **Output tab** and find the **Plot panel.** From there, simply click on the **Preview button**.

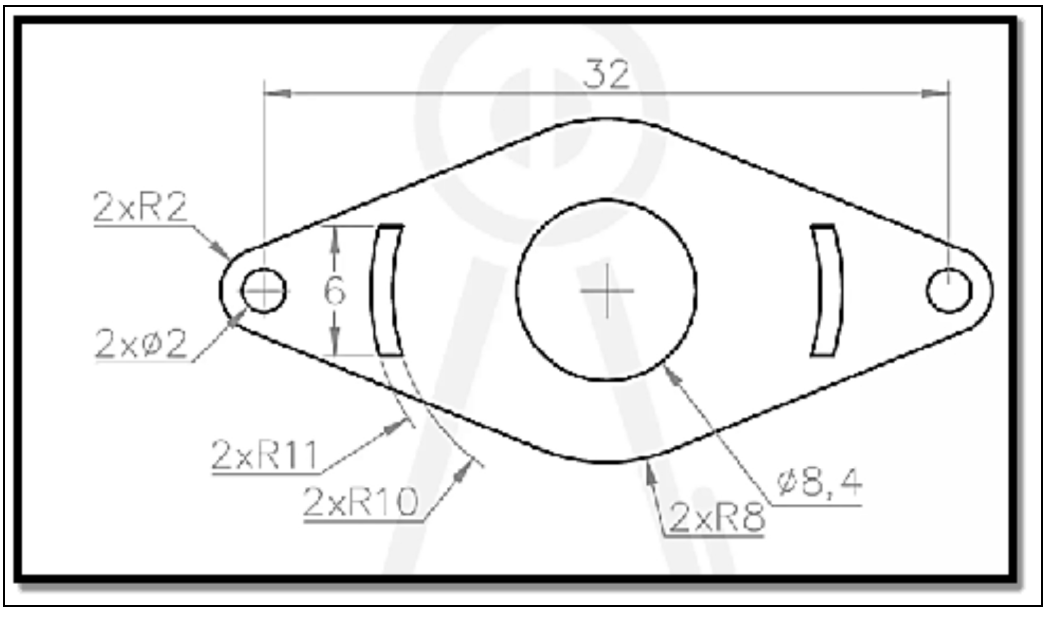
[OceanofPDF.com](OceanofPDF.com)

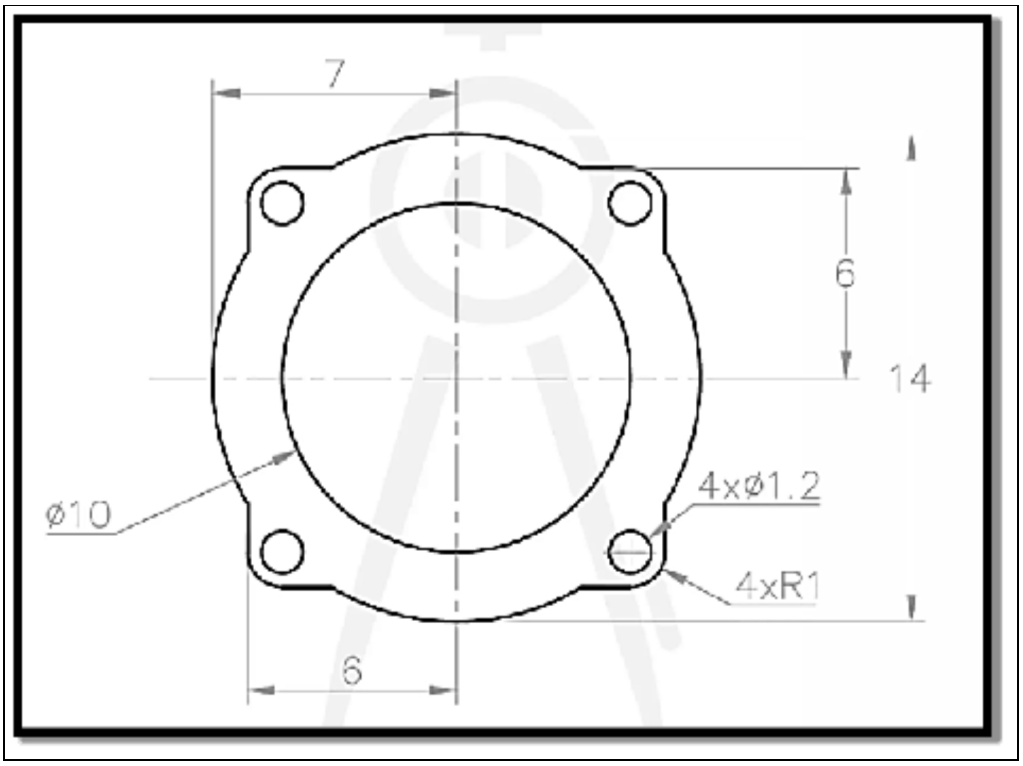
# 2D Practice Drawings

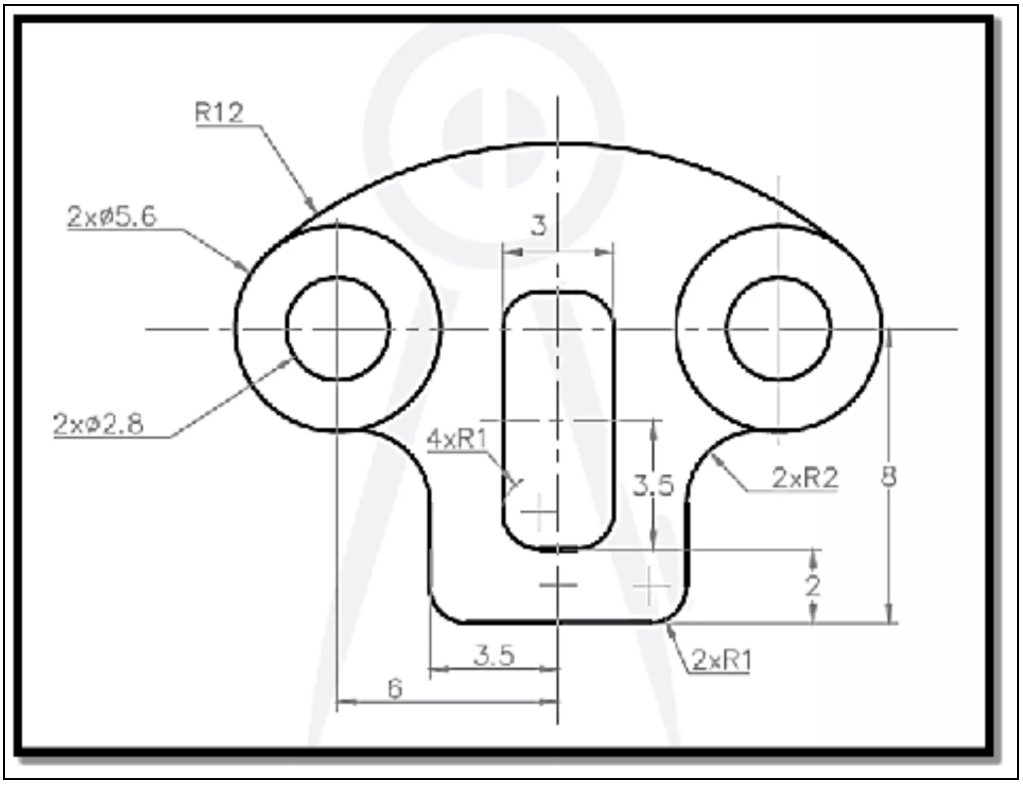


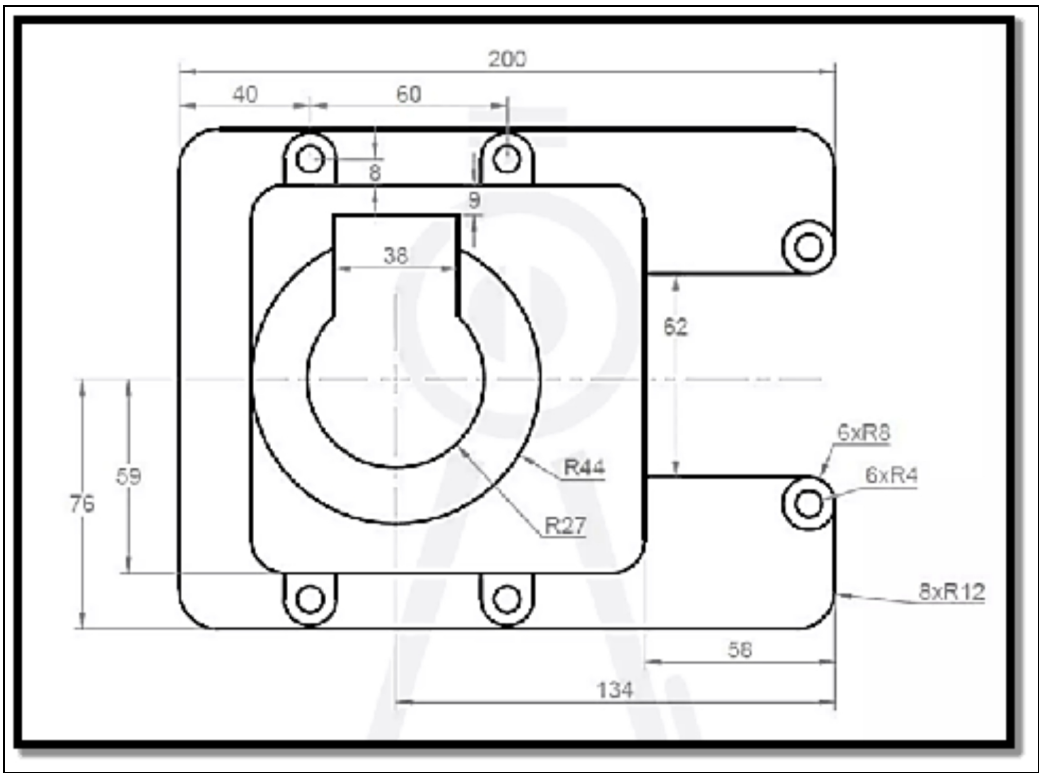



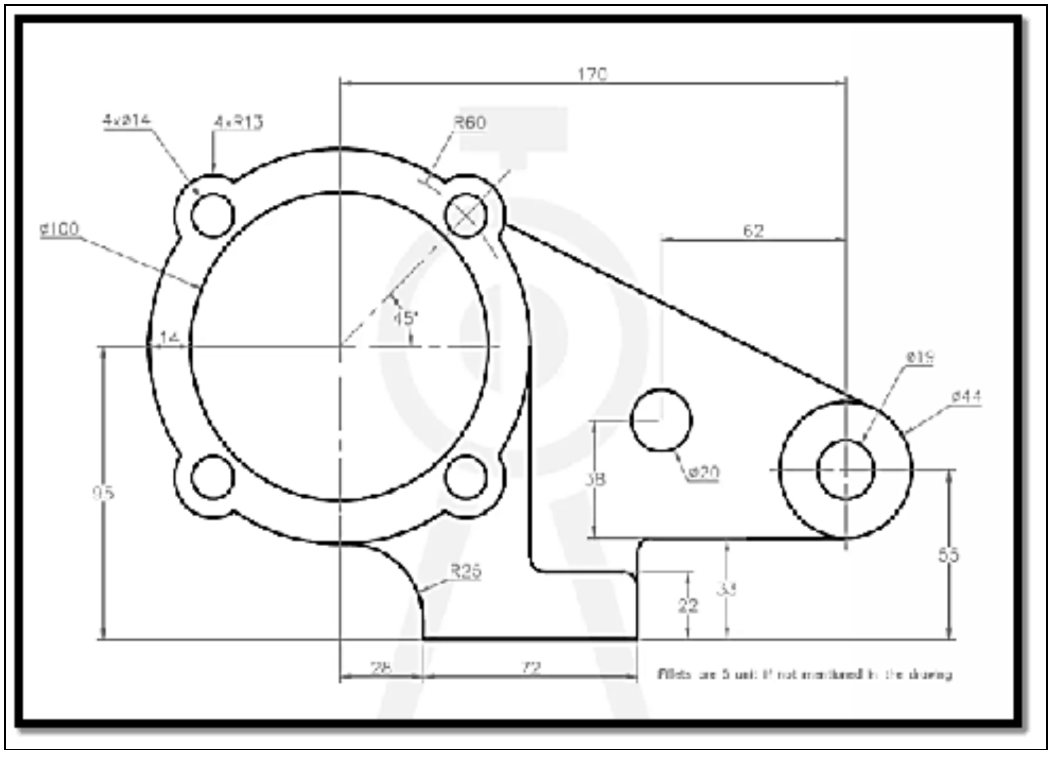

ø8.2
ø4.8
ø3.6
R5.7
18
2xR1.6
1.6
ø2.4
4.8
4
ø1.52
1
2.5
2
12

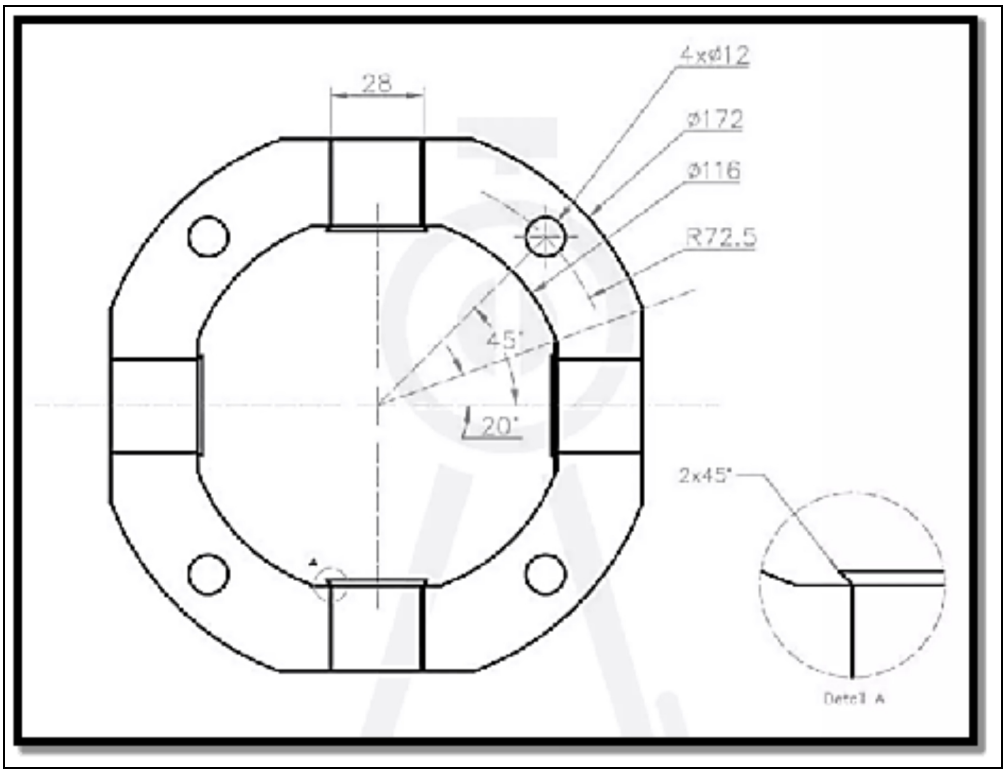

32
2xR2
6
2xø2
2xR11
2xR10
ø8,4
2xR8

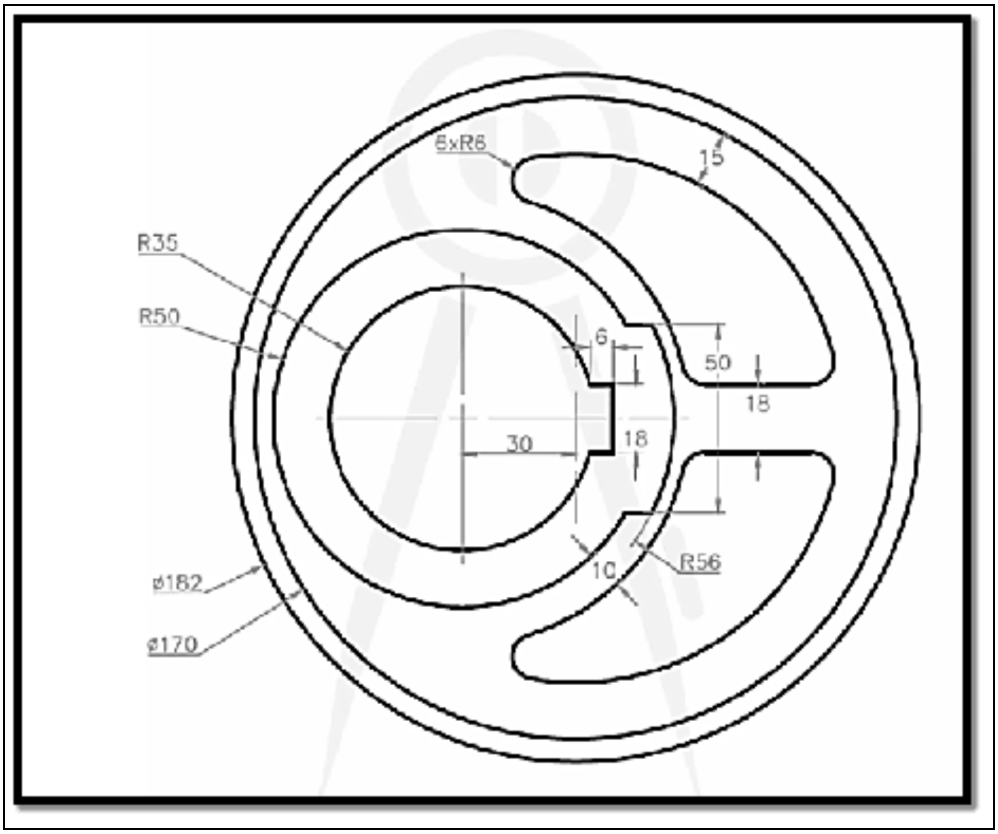

7
6
14
ø10
4xø1.2
4xR1
6

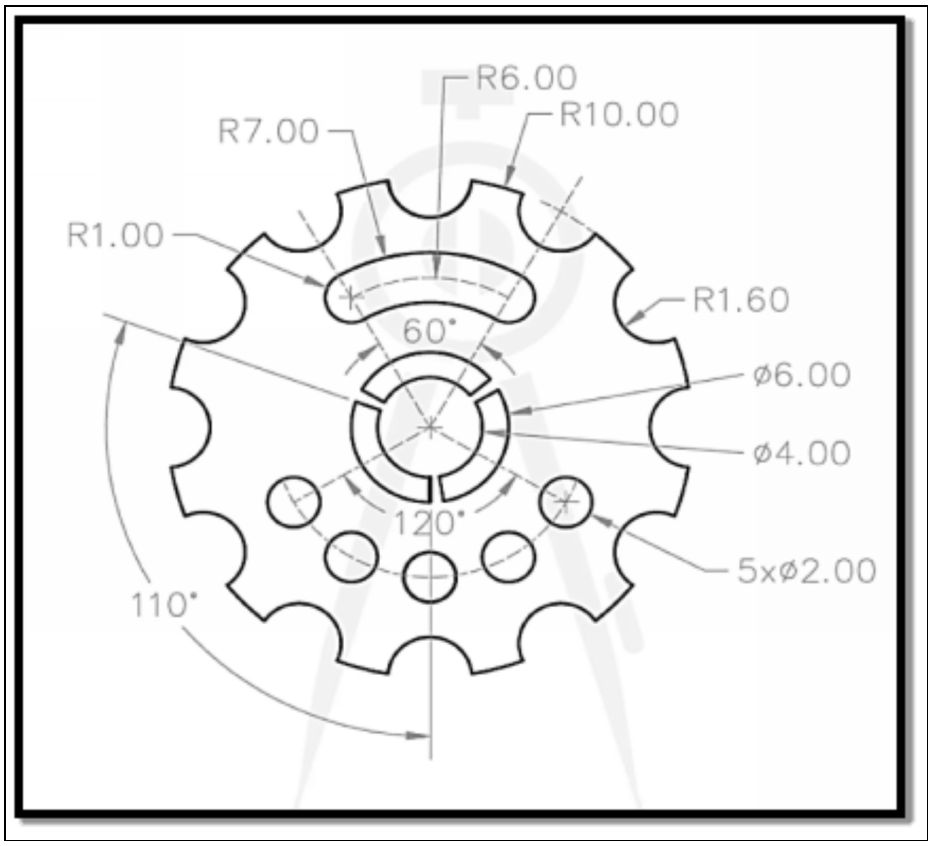

R12
2xø5.6
3
2xø2.8
4xR1
3.5
2xR2
8
2
3.5
2xR1
6

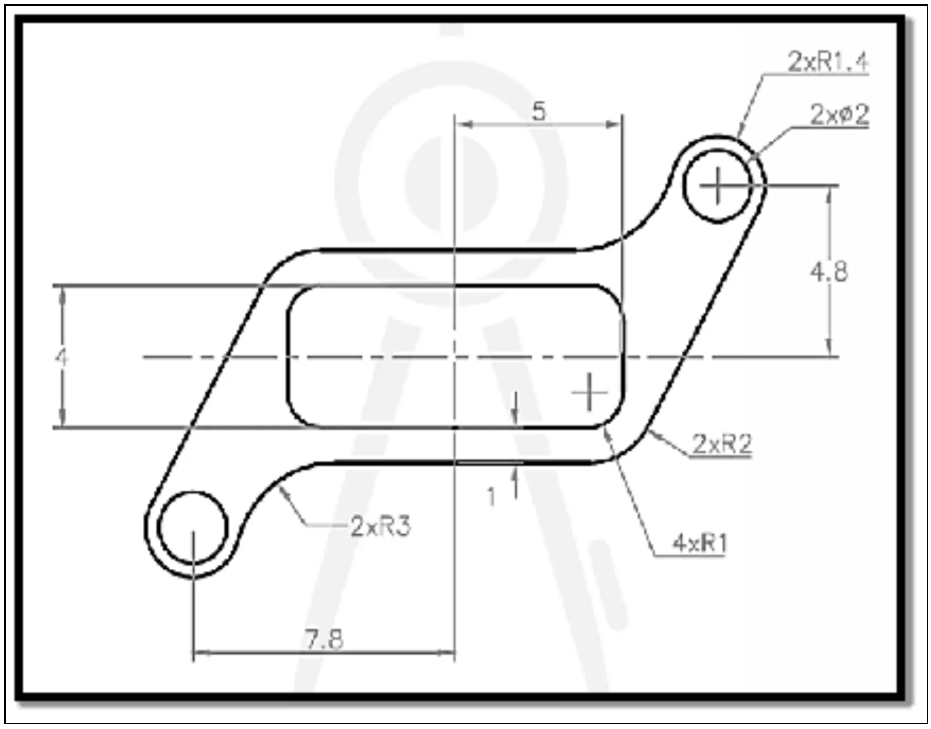

200
40
60
8
9
38
62
6xR8
6xR4
R44
R27
59
76
8xR12
58
134

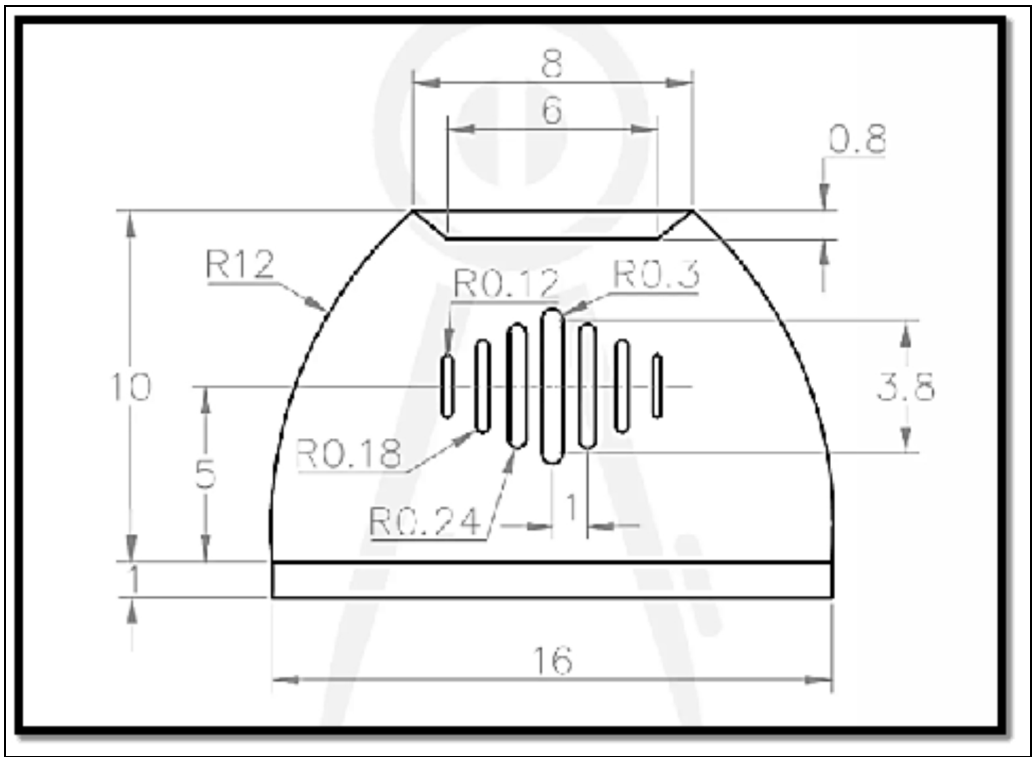

170
4xØ14
4xR13
R60
Ø100
62
15°
14
Ø15
Ø44
Ø20
38
95
55
R25
22
34
28
72

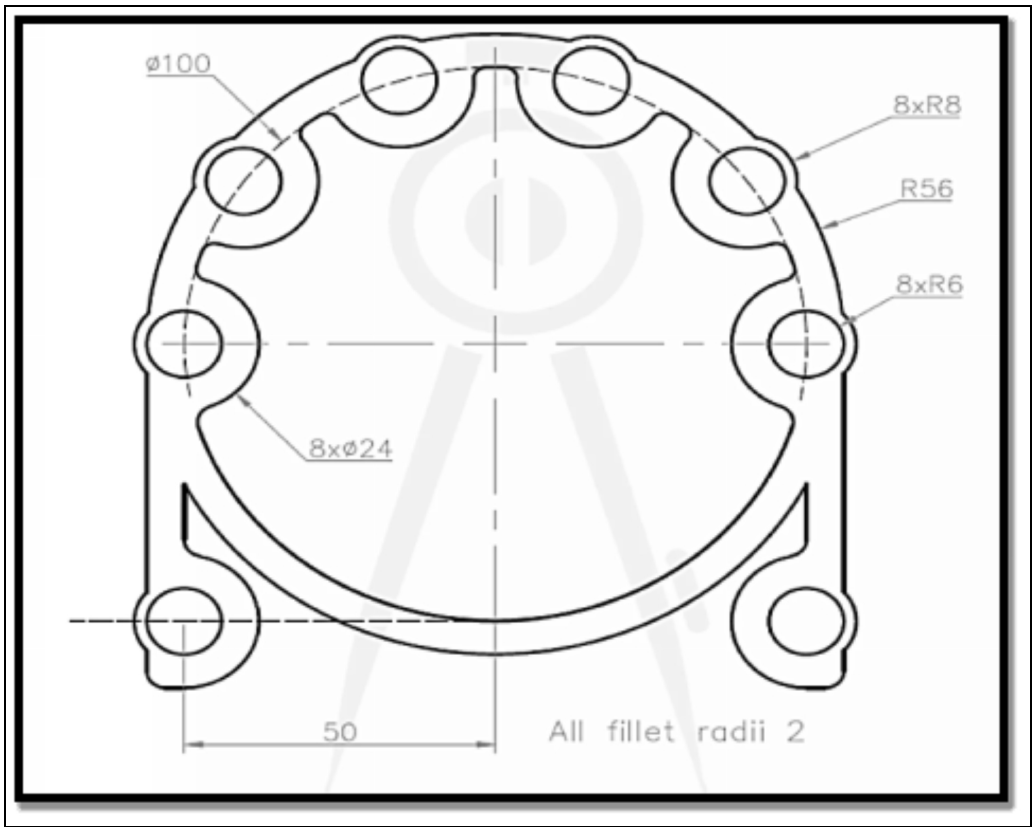

4xø12
28
ø172
ø116
R72.5
45°
20°
2x45°
Detail A

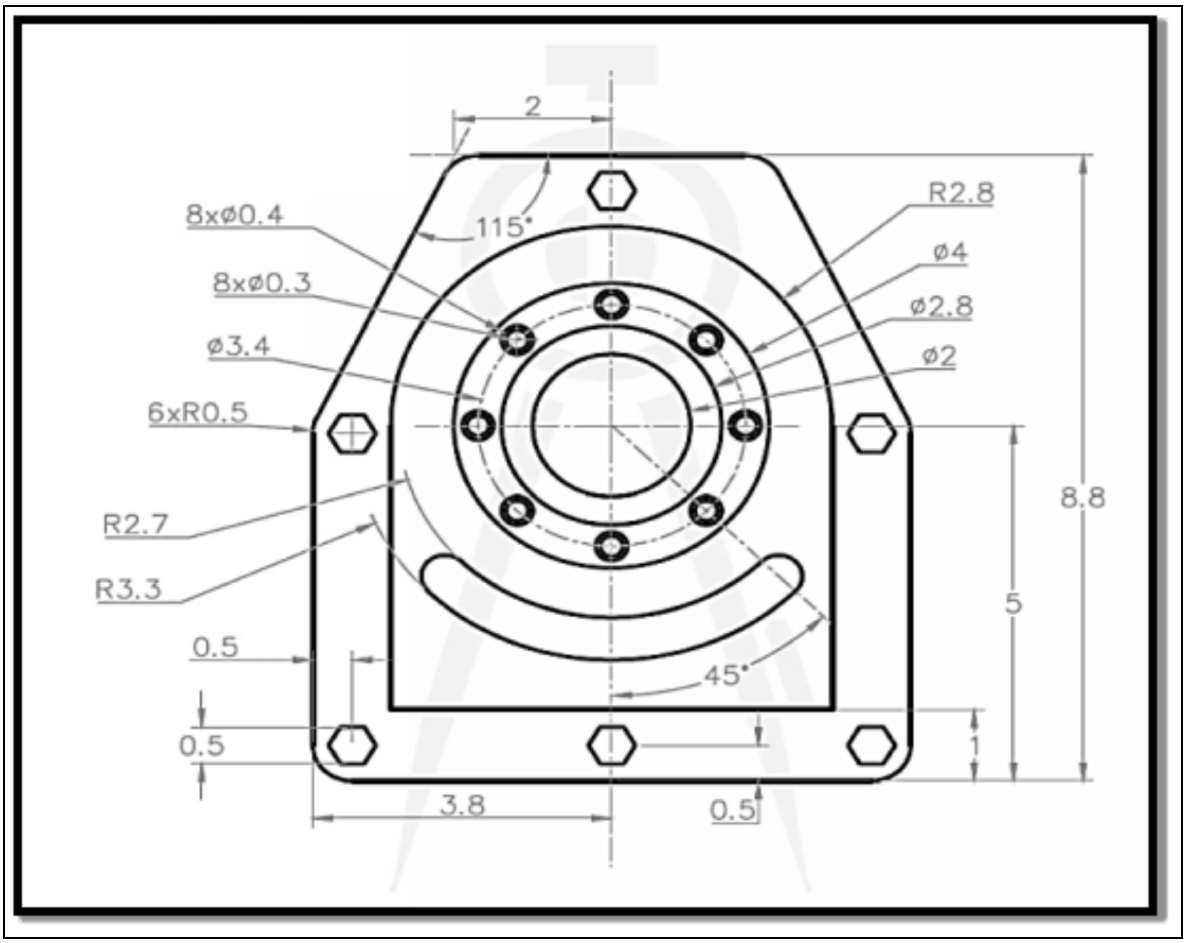

6xR6
15
R35
R50
6
50
18
30
18
10
R56
ø182
ø170

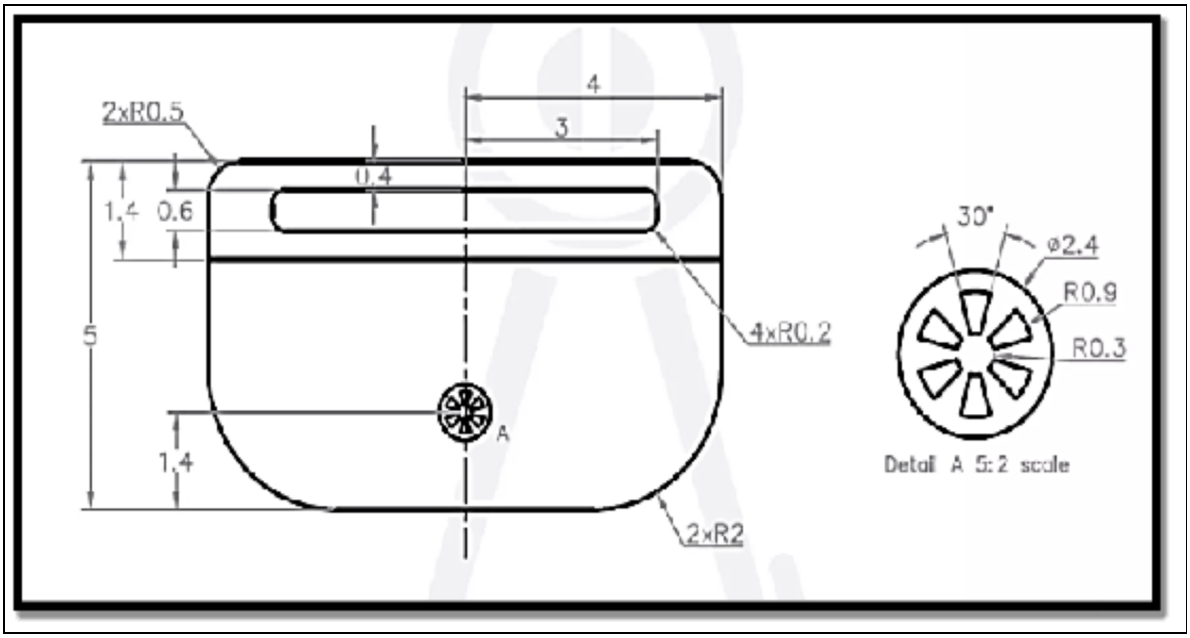

R6.00
R7.00
R10.00
R1.00
R1.60
60°
ø6.00
ø4.00
120°
5xø2.00
110°

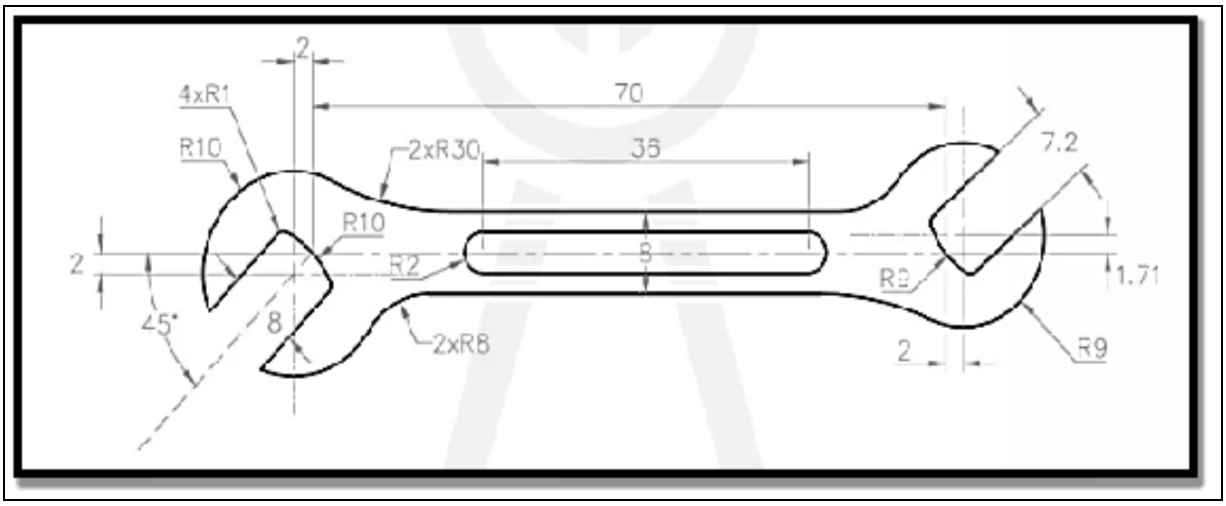

2xR1.4
2xø2
5
4.8
4
2xR2
1
2xR3
4xR1
7.8

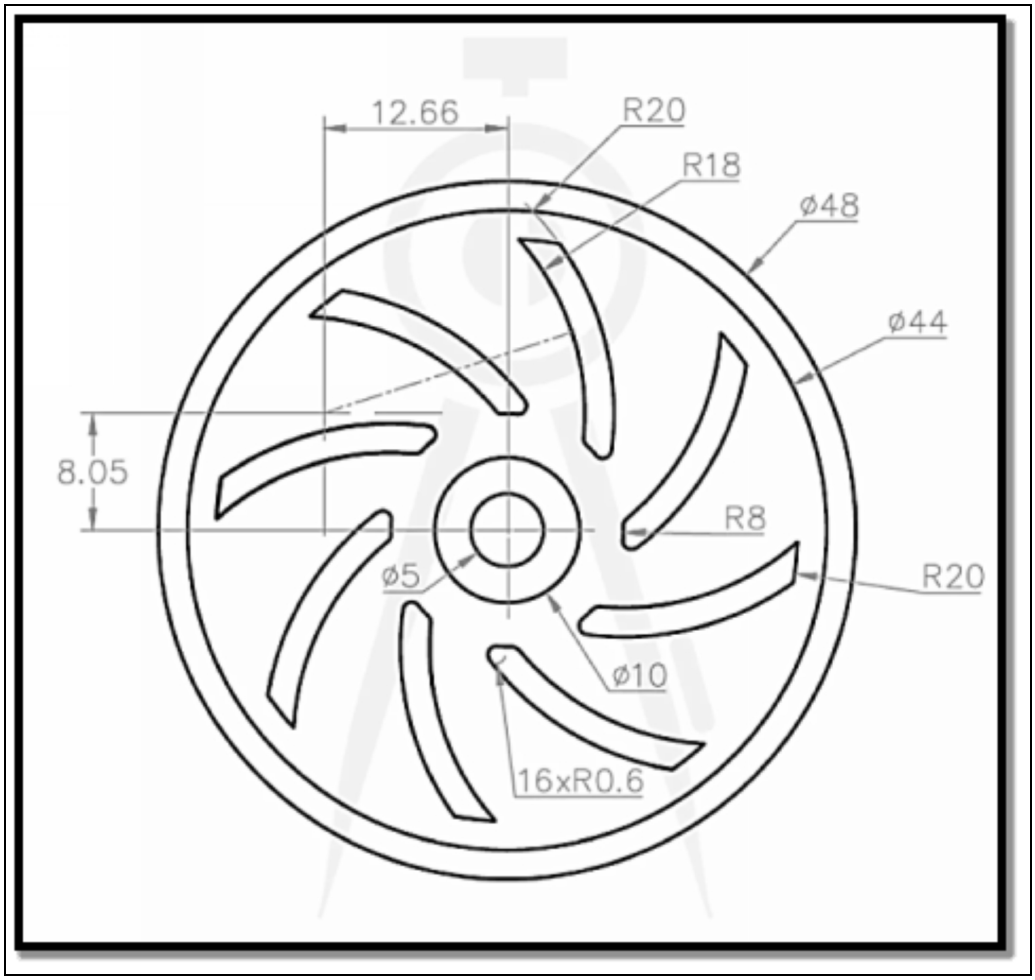

8
6
0.8
R12
R0.12
R0.3
10
3.8
5
R0.18
R0.24
1
1
16

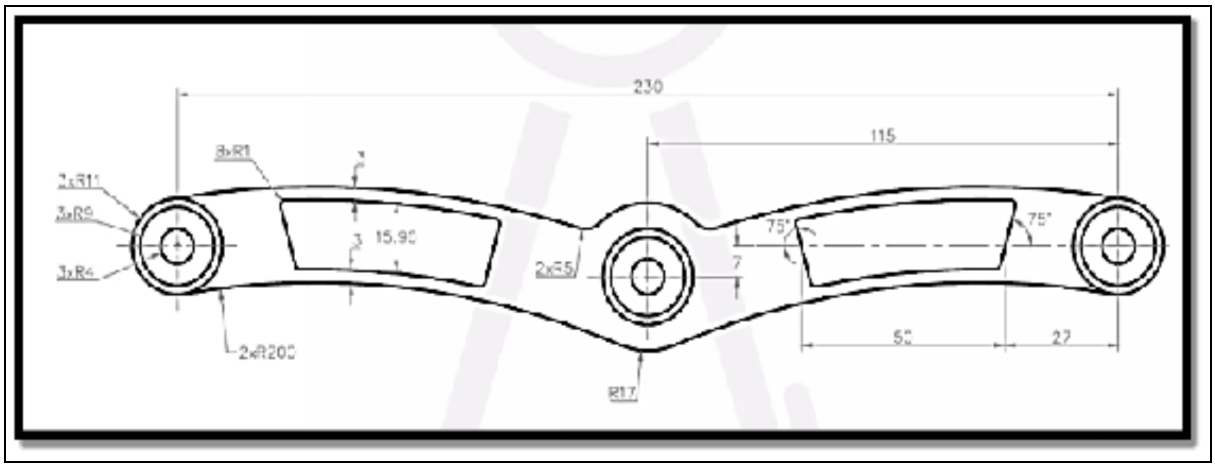

ø100
8xR8
R56
8xR6
8xø24
50
All fillet radii 2

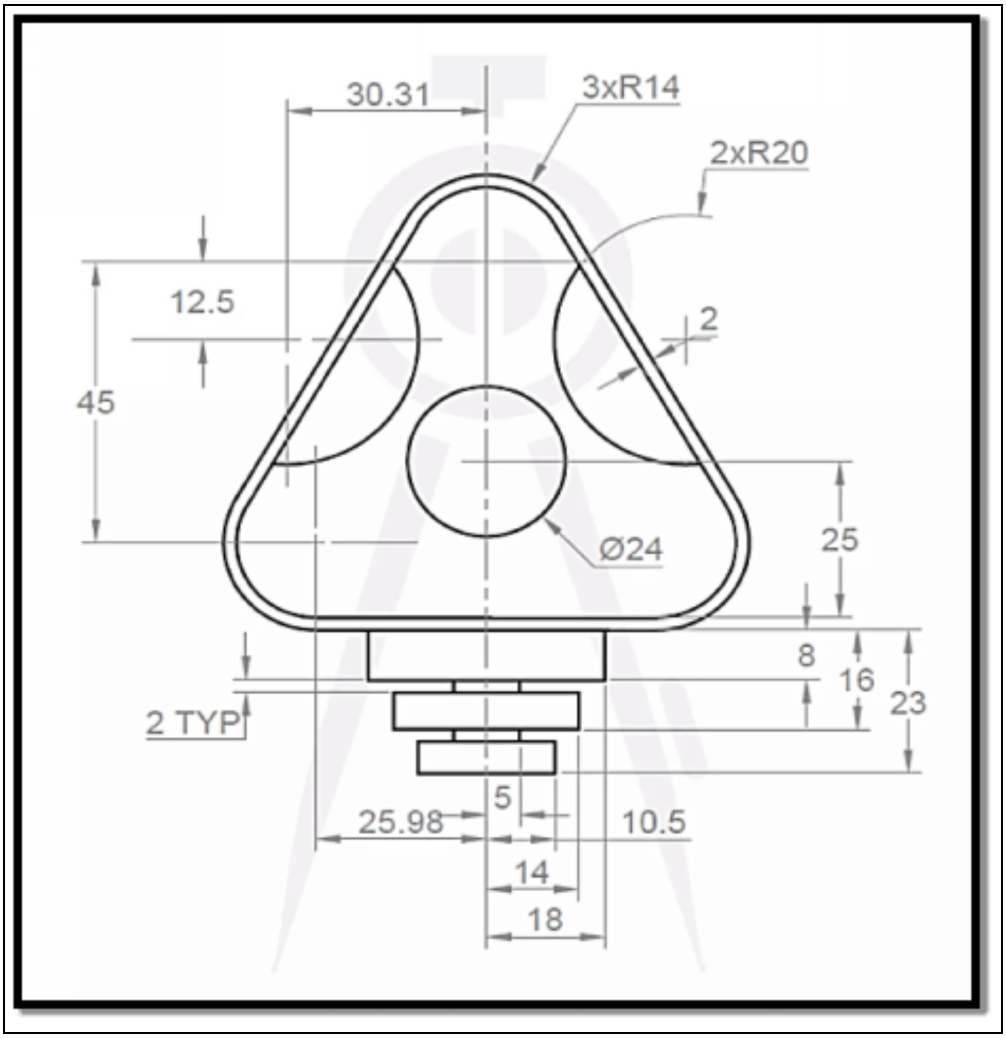

2
115°
8xø0.4
8xø0.3
ø3.4
6xR0.5
R2.7
R3.3
0.5
0.5
3.8
0.5
45°
R2.8
ø4
ø2.8
ø2
8.8
5
1


4
3
2xR0.5
0.4
1.4
0.6
5
4xR0.2
A
1.4
2xR2
30°
ø2.4
R0.9
R0.3
Detail A 5:2 scale


2
4xR1
70
R10
2xR30
35
7.2
R10
2
R2
5
R3
1.71
45°
8
2xR8
2
R9


12.66
R20
R18
ø48
ø44
8.05
R8
ø5
R20
ø10
16xR0.6


230
115
8xR1
3xR11
3xR9
3xR4
15.90
2xR5
75°
75°
7
50
27
2xR200
R17


30.31
3xR14
2xR20
12.5
2
45
Ø24
25
8
16
23
2 TYP
5
25.98
10.5
14
18

# INDEX

## 3

3D drafts and designs. , 5

## A

**A dialog box will be displayed for your reference** , 19, 21, 294, 297
**A prompt will appear** , 42, 92, 300
**A prompt will be displayed** , 78, 82, 228, 300
A selection set is a group of objects that have been chosen in AutoCAD for further editing or manipulation , 137
About AutoCAD 2025, 8
About AutoCAD 2025 Interface, 8
About Offsetting Objects, 75
Access an existing file , 6
access the additional options , 12
Access the AutoCAD web app , 23
access the command , 22, 26, 42
access the subsequent menu , 17
accuracy and adaptability , 1
accurate alignment of the graphics , 124
activate **OTRACK** , 40
add or remove commands , 9
**Add Selected** , 109
Add-a-Plotter Wizard, 147
Additional Angles, 32
additional buttons , 11, 17, 283
additional object snaps , 33
Additionally, you will acquire the skills to efficiently duplicate, relocate, and arrange objects , 129
**ADDSELECTED command** , 109
Adjust the cursor , 112, 145
adjust the **DIMCEN system** variable , 153, 155
Adjust the position of the circle by moving it diagonally in any desired direction , 139
ADJUSTING COMMANDS, 53, 75
adjusting the snap settings , 124
**Advanced 3D Modeling Tools** , 3
advanced automation possibilities , 1
advanced capabilities , 4, 75
**Advanced DWG Compare** , 3
**Advanced Mesh Modeling** , 3
AEC industry , 2
**After activation, when selecting an object, the following may be displayed** , 59

**After launching the program, a prompt will be** , 65
**After opening the program, a series of prompts will be displayed** , 64, 98
Aligning objects with the path , 92
alternate units , 161
Alternate Units , 162
An alternative approach to the arc , 43
An array is a pattern of objects that can be rectangular or circular, or even follow a specific path. There are three different types of arrays in AutoCAD , 141
An array is a way to represent an object by repeating it in a matrix format , 141
angle of the arc , 43
Angles in AutoCAD are measured in a full 360° rotation , 30
Annotative objects , 159
**Annotative option** , 163
Annotative style , 159, 163, 203, 204, 261
announcements from AutoCAD regarding your software or hardware. , 6
ANSI standard refers to a variety of guidelines and standards that have been developed and disseminated by the American , 133
applicable to 3D designs , 90
Application Menu, 8, 10, 21, 22, 23, 24
**Application menu houses a variety of commands** , 10
**Apply to Layout button** , 122, 308
architects can now bring their designs to life , 4
Architectural , 103, 104, 130, 149, 156
architectural drawing , 156, 213
architecture , 1, 5
**Architecture** , 4
arc's center point , 43
Arranging Objects, 88, 91
Arranging Objects – Path Array, 91
Arranging Objects – Polar Array, 88
**Array command** , 75, 88, 157
**ARRAY command** , 154, 283
Arraying Objects, 84
Arraying Objects – Rectangular Array, 84
**ARRAYRECT command** , 141, 142, 155
ASsociative , 85, 92
AUTOCAD 2025, 2
AutoCAD 2025 brings forth a multitude of fresh features , 3
AutoCAD 2025 enable planners to incorporate highly detailed and accurate information into their maps , 5
AutoCAD 2025 has been enhanced to deliver faster , 3
AutoCAD 2025 has expanded its support for automation , 4
**AutoCAD 2025 interface will be displayed** , 6
**AutoCAD 2025 is an incredibly versatile tool that is used in various industries. Several industries greatly benefit from the advanced features of AutoCAD** , 4
AutoCAD 2025 is well-positioned , 1

**AutoCAD 2025 now allows users to conveniently save and access their drawings using the AutoCAD web and mobile apps** , 23
**AutoCAD 2025 now offers a convenient "Share" feature that allows for seamless collaboration** , 23
AutoCAD 2025 offers a wide range of options and settings , 22
**AutoCAD 2025 offers the option to save drawings in various DWG file formats, ensuring seamless compatibility with previous versions of AutoCAD** , 22
AutoCAD 2025, created by Autodesk , 2
AutoCAD 2025, was created by Autodesk , 1
AutoCAD 2025, was created by Autodesk and represents the most recent breakthrough in the field , 1
AutoCAD displays a distinct menu , 74
AutoCAD displays coordinates , 16
AutoCAD displays precise rounded coordinate values , 124
AutoCAD Drafting, 27
AutoCAD drawing files , 20
AutoCAD expert , 25, 149, 156, 162, 274
AutoCAD has built upon its predecessor , 2
AutoCAD has consistently maintained the largest user base in the CAD industry , 5
AutoCAD has consolidated all the options , 13
AutoCAD has undergone numerous updates by Autodesk , 2
AutoCAD is a powerful tool that interior designers rely on to meticulously craft layouts , 5
AutoCAD is a versatile software , 5
AutoCAD LT is a specialized version , 6
AutoCAD offers a wide range of commands and features , 163
AutoCAD program comes with a range of templates that feature standard sheet sizes, complete with title blocks, borders, and predefined plot styles , 132
**AutoCAD prompts** , 76, 78, 174, 176, 190, 195, 208, 244, 259
AutoCAD provides a precise method for specifying points , 33
AutoCAD provides options for managing situations , 162
**AutoCAD provides prompts** , 120, 150, 169, 174, 175, 179, 181, 189, 243
AutoCAD provides real-time feedback in the command line , 137
AutoCAD remains the top choice for Autodesk , 5
AutoCAD revolutionized the field of drafting , 2
AutoCAD that encompasses the virtual environment where users can effortlessly create and modify objects , 129
AutoCAD versions , 22
AutoCAD was first introduced in 1982 by Autodesk , 5
AutoCAD Web , 3, 9, 23
**AutoCAD Web App Improvements** , 3
AutoCAD will display , 13, 28, 92, 139, 307
AutoCAD will present a list of available editing modes , 144
AutoCAD will provide you with the current line width , 43
AutoCAD. These include creating a new file, opening an existing file, saving the current file, saving the current file under a new name and in a different folder, importing from different file formats, exporting the current file to a different file format, printing and publishing the current file, and more , 10
Autodesk Account , 15

Autodesk continuously improves AutoCAD , 2
Autodesk Docs , 23
automatically configured with a spacing , 99
automation and scripting features , 5
AutoSave, 22
Axis Angle , 87, 144

## B

Backup files, 23
Basepoint , 60, 61, 62, 63, 73, 85, 92, 136
**Batch Plot command** , 307, 308
Begin a new drawing or access additional templates online. , 6
**begin a new project or access a previously created file** , 6
boost productivity , 3
boosting efficiency. , 5
both 2D and 3D designs within AutoCAD , 33
**Break command** , 75, 97
Building Information Modeling , 5
**By clicking the small triangle located at the right of the button, a menu will be displayed** , 31, 34

## C

capture the endpoint of a line or arc , 33
capture the perpendicular point on an object , 33
catering to both precise 2D drafting and 3D design. , 16
cause AutoCAD to separate the Page , 308
**Center** , 38, 39, 106, 107, 108, 109, 124, 150, 217, 228, 243, 256
center of the display , 164
center point and diameter method , 124
**Center Wheel [BASIC]** , 124
centerline dimension , 155
**CENTERMARK** , 150
**Chamfer command** , 75, 79
Chamfering Using the Distance Option, 79
Chamfering with Distance and Angle, 80
Chapter Drawing Projects, 122, 151, 209, 277
Choose a polyline or select multiple options , 45
Choose an object or indicate two origin points , 164
Choose the **AutoCAD Drawing Template** , 133
Choose the circle , 135, 137, 139, 141, 150, 206, 209, 281
Choose two circles from the options provided , 135
**Choosing options from the command line** , 100
Circle tool – drop-down menu, 108
CIRCLES AND DRAWING AIDS, 99
**clicking on a rectangular array** , 88

close a file in AutoCAD , 24
Close and save the file. , 48, 49, 51, 52
cloud integration , 1, 5, 23
**Cloud Integration** , 3, 23
**Cloud Integration and Collaboration** , 3, 23
Column Count , 87, 144
COLumns , 85
Columns panel , 142, 151
Command window , 8
Command Window, 15
command window in AutoCAD , 15
command window of AutoCAD , 15
commercial buildings , 4
Compatibility with different file formats, 22
completing your task , 20, 45, 176
complex design capabilities , 1
comprehensive features , 5
comprehensive help information , 13
**comprehensive list of commands** , 25, 29
comprehensive understanding of the different techniques , 21
**considering the process of arranging an object along a path** , 92
constructing a bridge , 104
**Construction** , 5
construction professionals , 2, 5
containing and crossing features , 64
**context tab called Array will appear** , 91
CONVERTING AND DRAWING POLYLINES, 51
coordinates of points , 36
COPIES, TEMPLATES, AND ARRAYS, 129
**COPY command** , 134, 135, 151, 154
Copying with Grips, 135
COUnt , 85
count in 3D designs , 86
craft intricate and comprehensive depictions , 3
Create a circle with a radius of 1 and position it near the center of the drawing area at coordinates (9,6 , 137
create a triangle , 164
create highly detailed floor plans , 4
Create line segments and arc segments in AutoCAD , 42
Creating a new dimension style , 161
Creating circles in AutoCAD is a breeze , 36
creating detailed drawings and designs for various purposes , 5
Creating Linear Dimensions in AutoCAD, 163
Creating the layers Dim, Hatch, and Text, even if they are not currently used, can be a time-saving and beneficial practice for your template in the future , 131
Crosshairs , 8
Crossing Mode (C), 54

Crossing Polygon Mode (CP, 54
crossing-polygon , 64
Current chamfer , 80
**Current dimension style** , 163
Current file name , 8
Current settings , 61, 76, 78, 82
current version of the drawing , 9
**Customization and User Interface** , 4
customizing the way your files are saved , 22
cuTting edges , 82

## D

**Dark Mode Advancements** , 4
dealing with an arc , 69
Decimal , 104, 105, 161
Decimal Degrees , 161
decimal places , 103, 105, 124, 158, 161, 162, 163, 214
decipher AutoCAD's instructions , 15
default choice , 144
delete an object in AutoCAD , 58
demand precision and dedication , 21
Description box , 160, 163
**designate a new base point** , 93
designate points using OTRACK , 42
designed templates , 20
designing a wide range of structures , 4
desired location for the dimension line , 164, 166, 167, 182
desktop application , 2
detailed drawings and designs , 5
diameter symbol , 125, 182, 212
different configurations , 102, 161
Different Object Snaps, 39
Different Object Snaps Linked to Circles and Arcs, 39
different precision , 163
**DIMALIGNED** , 164, 167
**DIMANGULAR** , 164, 178
dimension lines and symbols , 161
dimension settings , 158, 161
Dimension specifications , 163
dimension units and drawing units , 161
dimension your drawings , 158
Direct any feedback you may have to Autodesk. , 6
disable Ortho in AutoCAD , 31
Disable **Ortho Mode** if it is currently enabled. , 138
disable Polar Tracking , 31
Displacement , 60, 61, 64, 140

**Displayed on the lower left corner of the screen are the model tab and the layouts, as illustrated below** , 18
Distance (two distances) , 79
Distance and Angle , 79, 80, 81
Distance Entry method , 48
distinct manners , 144
downward direction , 14, 263, 265
draw a line , 18, 65, 111
**Draw lines in the drawing** , 49
DRAWING LINES USING DYNAMIC INPUT, 48
DRAWING PROJECTS, 48
drawing quite straightforward , 125
**DRAWING USING OSNAP AND OTRACK** , 49
Drawing1.dwg , 20
dropdown menu. , 162, 183
Dynamic Input , 28, 29, 30, 31, 48, 68, 108, 110, 118, 243
**Dynamic Input offers a range of functions** , 28

## E

**Edit Source command** , 88
effortlessly create precise and flexible models , 5
electrical systems , 4
eliminating the need for manual techniques , 2
empowering designers , 2
empowering drafters , 2
Enable Snap Mode and Grid Display , 99
Enable **Snap Mode** if it is currently disabled , 138
enable snapping , 125
**Endpoint** , 33, 49
endpoints of an arc display , 69
engineering , 1, 4, 5, 104
**Engineering** , 4, 104
engineers , 2, 5
**Enhanced Annotation Tools** , 4
**Enhanced File Handling** , 3
enhanced performance , 3
enhancements , 2, 3
enhancing communication , 5
**Enhancing Visuals and Delivering Presentations** , 4
Enter "**Limits**" at the command line. Provide the coordinates for the lower-left corner. Provide the coordinates for the upper-right corner , 129
Enter all <**Enter**>. All circles will be displayed in a gray color , 141
Enter new value for MIRRTEXT , 65
Enter the x and y coordinates, using the [**Tab**] key to navigate between the two fields , 30
era of BIM , 5

**ERASE command** , 99, 107, 113, 114, 115, 141
**ERASE command** and restore the circles , 114
Erase source objects? , 65
Essential Techniques, 21
Essential Techniques for Saving in AutoCAD, 21
essential tool for architects , 2
essential tools. , 15, 169
establishing a new benchmark , 1
ever-evolving requirements , 1
Exact Angles, 31
Exact Angles (Ortho vs. Polar Tracking, 31
EXACT ANGLES PRACTICE, 48
**examine how AutoCAD will handle the given scenario** , 93
**Example of a crossing** , 54
exceptional precision , 4
Exercises to Enhance Line Drawing Skills with Dynamic Input, 29
exit the ARRAYRECT command , 143, 144
exit the Dimension Style , 163
exit the LINE command , 103
**Extend command** , 75
extended period , 13
extension *.dwt. , 20
extension lines. , 158, 164, 165, 166, 179

## F

facilitating more informed decision-making , 5
**Fan Bezel [INTERMEDIATE]** , 125
Fence Mode (F), 56
File tab , 8, 19, 24
**Fillet command** , 75, 77, 79, 81
find the **Plot panel** , 307, 308
**First, you'll find a comprehensive list of commands. From there, simply choose the desired group and proceed to undo them** , 25
five more columns , 145
**Floor Framing** , 156
**Floor Framing [ADVANCED]** , 156
Fractional , 104
fundamental , 1, 104, 129, 146, 158, 194, 268
further manipulate your lines , 68

## G

gaining a comprehensive understanding of the proposed layouts , 5
GETTING STARTED WITH AUTOCAD, 2
go to the Grid Behavior panel and deselect the option , 130
goals of enhancing the user experience , 1

Google Drive , 23
Gradually glide the cursor across one of the grips. When performing this task with precision, it becomes apparent that the cursor seamlessly attaches to the grip as it glides over it. When the cursor is locked onto the grip, the grip will change its color to orange. When you're positioned on any of the quadrant grips, you'll notice a value of 1.00 displayed, which represents the circle's radius , 140
graphical area , 14, 15, 16, 24, 28, 31
Graphical area , 8
Graphical Area, 8, 16
**graphical representations of each OSNAP** , 40
grasp the significance of AutoCAD 2025 , 2
greater collaboration , 1
GRID AND SNAP, 52
Grid spacing is set to 0.50 units and the coordinate display is turned on. Snap is set to 0.25 and LTSCALE is set to 0.5 , 131
***Grill* [BASIC]** , 153

## H

handling intricate geometries , 4
**Here is a step-by-step guide on how to create a template** , 131
**Here is a visual representation of the right-click menu** , 28
horizontal line of length , 164
horizontal lines , 16, 17, 40
How to Adjust Plot Settings, 146
How to Adjust Units, 103
How to Chamfer Objects, 79
How to Change the Grid Setting, 99
How to Change the Snap Setting, 100
How to Close Drawing Files, 24
How to Convert Polylines to Lines and Arcs, 44
How to Copy Command, 60
How to Create a New AutoCAD Drawing, 18
How to Create and Save a Dimension Style, 158
How to Create and Save a Drawing Template, 131
How to Customize Your Workspace, 115
How to Draw Circles, 36, 106, 107
How to Draw Circles by Specifying a Center Point and a Diameter, 106
How to Draw Circles by Specifying a Center Point and a Radius, 107
How to Draw Circles Using Circle Command, 36
How to Draw Circular Ars Using Arc Command, 38
How to Draw Lines, 27, 42
How to Draw Lines and Ars Using Polyline Command, 42
How to Draw Lines Using the Line Command, 27
How to Edit a Rectangular Array, 86, 88
How to Edit a Rectangular Array Using Grips, 86
How to Edit a Rectangular Array Using Quick Properties, 88
How to Edit a Rectangular Array Using the Context Tab, 88

How to Enable Running OSNAPs, 34
How to Extend Objects, 82
How to Fillet Objects, 77
How to Join Objects, 66
How to Lengthen Objects, 67
How to Offset Using the Offset Distance Option, 76
How to Offset Using the Through Option, 76
How to Open an Existing AutoCAD Drawing, 20
How to Plot or Print a Drawing, 119
How to Save a Drawing Template, 131
How to Save Your Drawings, 21
How to Set Drawing Limits, 129
How to start using AutoCAD, 5
How to Trim Objects, 81
How to Use Break Command, 97
How to Use Erase Command, 57
How to Use ERASE Command, 113
How to Use Grips, 68, 70
How to Use Grips and Dynamic Input, 68
How to Use Grips to Edit Objects, 70
How to Use Mirror Command, 65
How to Use Move Command, 59
How to Use Object Snap Tracking with OSNAP, 40
How to Use Polar Snap, 47
How to Use RECTANG Command, 118
How to Use Rotate Command, 62
How to Use Scale Command, 63
How to Use Selection Cycling, 58
How to Use Single-Point Object Snap, 109
How to Use Snap and Grid to Specify Precise Points, 45
How to Use Strech Command, 64
How to Use the Array Creation Context Tab, 85
How to Use the ARRAYRECT Command, 141
How to Use the Copy Command, 134
How to Use the MOVE Command, 137
How to Use the Multiple Option, 77

## I

If 1B does not appear at the top of your list, it could be due to the sorting order of your list. Make sure to check the PTW Templates and Sheet Sets folders as well. To change the order, simply click on the Name column , 134
If you're tired of repeatedly setting up layers, units, grids, snaps, and scales for every new drawing, then keep reading , 129
Illustration of Specifying Lengths, 30
Illustration of Specifying Lengths and Angles, 30
immersive and captivating experience , 4
impact of different modifications , 146
implementation of expanded 3D modeling capabilities , 1

Important, 122, 143, 156, 169, 171, 233
improved file handling capabilities , 3
**Improved LISP and Python Support** , 4
**Improved Performance and** , 3
In the Array Creation contextual tab, simply click on the **Columns edit box** and enter the value of 5, then press **Enter**. Pressing <**Enter**> will automatically add a column, resulting in a total of 5 columns , 143
inches and centimeters , 163
incredibly efficient tool , 70
Increment Angle, 32
Indicate a point that is 3.00 units to the right of your base point. After moving the circle, the rubber band and the original circle vanish , 138
Indicate a point to duplicate the two highlighted circles , 136
**Indicate the base point or provide the displacement value** , 138
Indicate the desired base point , 135, 139, 272
Indicate the desired base point. Similar to the MOVE command, AutoCAD will request input for a second point , 135
**Indicate the desired window for printing. Provide the coordinates for the first corner** , 120
Indicate the starting point , 42, 201, 240, 245
Info center , 8
InfoCenter, 15
information , 4, 5, 38, 43, 68, 78, 85, 108, 122, 140, 142, 176, 188, 266, 269
initiate Crossing mode , 53
initiate the ZOOM command , 99
Input a new value , 99
**Interior Design** , 5
**Intersection** , 33, 49
intricate connections , 3
intricate drawings , 3, 21
intricate grid , 124
INTRODUCTION, 1
It is important to observe that all endpoints of the lines align perfectly with the grid points , 128

## J

Join command , 66
joining arcs in AutoCAD , 67
joining arcs to create a single arc , 66
joining lines , 66
Joining lines and arcs, 44
Joining lines and arcs to form a polyline, 44
jump exact distances , 45

## L

Lasso Selection, 56
Launch AutoCAD 2025 , 48, 49, 51, 52

**launching AutoCAD** , 44
launching the program , 45, 61, 62, 63, 65, 97, 298
Layout tab , 8, 295, 296, 298, 299, 300, 306
Lengthen , 67, 72
Let's create a circle with a radius of 0.5, perfectly centered at the coordinates (2,2). , 141
line joined to source , 66
Linear dimensions panel , 161, 162
Linear dimensions panel. , 161
LISP and Python scripting capabilities , 4
locate the midpoint of a line or arc , 33
Locate the tangent point using object snap, 113
locating the **Object Snap Tracking button** , 41

## M

Make sure you're in the drawing from the previous exercise , 132
make well-informed decisions , 5
manufacturing , 5
**Manufacturing** , 5
Mastering Keyboard Shortcuts, 22
Measure or Divide , 92
mechanical components , 4
microcomputers equipped with internal graphics controllers , 2
**Midpoint** , 33, 49
Mirror , 65, 72
Model Space and Paper Space, 129, 286
Model tab , 8, 287
modify the default settings of Polar Tracking , 32
**Modify the existing drawing according to the following instructions** , 131
modifying 3D models , 3
modifying dimension variables , 162
more efficient working environment , 3
more precise snap spacing , 124
Moving with Grips, 139
multifunctional grip , 144
multiple files in AutoCAD , 21
multiple layouts in your drawing , 307
My Computer , 147
**MyStyle** , 161, 163

## N

Navigate to the Primary Units tab , 161
Navigation bar , 8
**Nearest** , 33, 152, 184, 187, 189, 190
**New Dimension Style dialog** , 160, 161, 162, 163
**New Machine Learning Tools** , 4

new parametric design features in AutoCAD 2025 , 5
newest version of a highly popular computer-aided design , 2
newest version of computer-aided design , 1
Notable Features, 3
Notable Features of AutoCAD 2025, 3
**Note** , 20, 132, 143, 195, 227, 232, 295
note of the diameters , 125
**notice the Array Creation context tab** , 92
Noun/Verb Editing, 114
numerous options available for certain commands , 13

## O

object selection , 99, 109, 113, 115, 118, 138, 141, 151, 165, 170, 208
Object Snap , 33, 34, 35, 40, 42, 48, 49, 110, 184, 187, 190
objects in various ways , 144
Observe the coordinate display , 124
Observe the dynamic input tooltip and extend the rubber band until the tooltip displays 3.00<0°. , 138
observe the issue firsthand. , 124
offering unparalleled flexibility to work from any location. , 3
**Offset button** , 75
**Offset command** , 75
OFFSETGAPTYPE , 76
**On the Start tab, you can achieve the same result by selecting the New drop-down menu, as demonstrated below** , 20
open an existing drawing file , 20
**Open button** , 20, 21, 134, 159
Open the "**Save Drawing As**" dialog box and choose "**Autodesk Docs**" from the "**Save in**" drop-down menu , 23
open the Drafting Settings , 46
**opening the command window** , 57
Opening the Dimension Style Manager , 159
optimal strategies , 21
optimize workflows , 3
**Optimized Rendering Engine** , 3
**OPTIONS** , 22
**options for selecting drawing orientation** , 148
options in AutoCAD , 74, 161, 166, 191
organized manner , 157
Ortho function , 31
OTRACK point , 42
overall scale, , 163

## P

Paper Size, 147

**Parametric Design Improvements** , 3
**Path Array button** , 91
Path Associative , 92
percentage length , 67
**Perform a variety of tasks without the need for manual input** , 53
perform specific actions , 15
**Perpendicular** , 33, 49
personal computers , 5
***Pin Trailer Wiring Diagram*** **[BASIC]** , 151
Plot Area, 120, 149
Plot Offset, 149
PLOT option , 149
Plot Preview, 121
Polar Angle Measurement, 32
**polar array command** , 91
Polar Associative , 89
**Polar command** , 42
**Polar Tracking** , 31, 32, 33, 47, 48, 51, 52, 184
**Polyline command is capable of performing a variety of tasks** , 42
portrait orientation , 148
positioning of dimension text , 162
powerful and versatile options , 3
powerful tool for creating intricate designs , 4
precise and accurate drawings of lines , 16
Precise Drafting, 33
Precise Drafting using Object Snap, 33
precise spacing and rotation , 61
pre-defined style in AutoCAD , 159
Press the "z" key on your keyboard and then hit the **Enter key** , 130
Press the <**Enter**> key or the spacebar to complete the selection of objects. AutoCAD will request the selection of a base point , 139
Press the Enter key once more to finalize the command , 141, 243, 244, 245, 249
Press the Enter key to exit the command. , 135, 171, 261
**Preview button** , 121, 149, 150, 308
preview your drawing outside , 308
Previous (P), Last (L), and more modes, 57
primary dimension unit , 162
primary units , 161
Primary Units , 161, 162
**procedure to add a plotter** , 147
Proceed to save and close. , 52
produce designs with unparalleled accuracy and efficiency , 2
professionals working in a wide range of sectors , 1
Provide an additional point to generate two additional copies , 136
provide precise measurements , 45
Provide the coordinates for the second point. A duplicate of the circle will be visible. The command line will display a prompt asking for the second point of displacement , 135

Provide the coordinates for the second point. Feel free to repeat this process as many times as you desire. Once you delve into this field, you'll experience the exhilaration of effortlessly conjuring rings , 135
Provide the values for the Snap X Spacing and Snap Y Spacing , 47

## Q

**Quadrant** , 39, 111, 178
**Quadrant option** , 111
Quick Access toolbar , 25, 26, 114, 116, 117, 119, 130, 132, 134, 146, 275
Quick Access Toolbar, 8, 18, 20
Quick Properties , 88, 91, 96, 192, 193, 249, 251, 254, 255, 256, 258
**Quick Properties panel** , 91, 96

## R

realm of design , 1
realm of design and drafting technology , 1
recommended settings , 156
RECTANG command , 99, 118, 119
Rectangular Associative , 85
Redo command, 26
Regularly saving your work is essential in any software , 21
**Replace Item command** , 88
Replace Items command , 88
respective lengths and angles , 68
revolutionized the way users work , 3
Ribbon , 4, 8, 261
Ribbons, 8, 11
Rotate , 62, 72, 174, 191, 281

## S

sacrificing accuracy , 70
Save Options and Settings, 22
Save the file by clicking on the Save button , 133
Saving the existing workspace, 117
Saving to Autodesk Docs, 23
saving your drawings , 21
Scale , 63, 72, 120, 163, 202, 203, 204, 205, 223, 232, 271, 272, 274, 303
Scientific , 104
**Scripting and Automation** , 4
seamless compatibility , 22
Select **1B** or the file name you have used from the list of templates , 134
Select a grip on one of the two circles that are highlighted. The color of the grip needs to be modified , 136
Select cutting edges , 82

Select lines to join to source , 66
Select object , 76, 82, 97, 98
Select object to offset , 76
Select objects, 53, 60, 61, 62, 63, 64, 65, 82, 84, 89, 92, 139, 195, 218, 237
Select Objects prompt , 54, 58
Select path curve , 92
Select source object or multiple objects to join at once , 66
Select the **Linear tool** , 164
select the **Plot command** , 307
Select the **Rectangular Array tool** located in the Array drop-down menu of the Modify panel on the Home tab of the ribbon , 141
selected the objects , 65
Selecting Objects in AutoCAD 2025, 53
Selecting Options, 99
Selecting Options from the Command Line, 99
selecting the drop-down list , 99
Selection Cycling button , 58
send the layout to the plotter , 308
series of new features and advancements , 1
Set up layers with corresponding colors, line types, and configurations. , 131
**Setting drawing limits is an essential skill in AutoCAD** , 129
Setting up a Printer, 120
setup from the current layout , 308
several aspects , 156
Share Drawing, 9
shared lines , 70
Sharing Drawings, 23
**Show Menu Bar** , 9, 116
Sign in , 8
simply click on the **Preview button** , 148, 308
smoother navigation , 3
SNAP (OSNAP) Objective, 49
**Snap Mode button** , 100, 101, 102, 130, 178, 276
**Snap Settings option** , 46, 47
Spacing , 47, 85, 87, 144, 297
specific prompt , 67
Specify base point , 60, 61, 62, 63, 64, 272
Specify base point or [Displacement , 60, 61, 64
Specify chamfer length , 80
Specify endpoint of line , 55
Specify first break point , 97
Specify first chamfer distance , 79
Specify first point of mirror line , 65
Specify next point , 27, 112
Specify rotation angle , 62
Specify second break point , 97
Specify second breakpoint , 97

Specify second chamfer distance , 79
Specify second point , 60, 61, 64, 65
Specify second point or <use first point as displacement , 60, 64
Specify the center point , 89, 243
Specify the drive and folder you want to use. , 21
Specify the first point , 27, 55
specify the second point , 61, 64
specifying the starting point , 30
specifying two points , 65, 138, 244, 274
**Standard** and **Annotative** styles , 159
Start tab , 8, 16, 20, 99, 130
Start Tab, 16
starting AutoCAD , 20
Status bar , 8, 58, 68, 303
Status Bar, 16
straightforward help feature , 13
**Streamlined Ribbon Interface** , 4
streamlining repetitive tasks , 5
Stretch , 64, 68, 69, 70, 72, 74, 140
Stretch and Lengthen , 68, 69
Stretch and Radius , 69
***Switch Plate* [INTERMEDIATE]** , 126

## T

**tabs and panels** , 4, 11
**Tangent** , 39, 92, 110, 112
Template file folder. When multiple users share a computer at different times, altering AutoCAD settings can lead to confusion. If your instructor or CAD manager , 132
Template files are typically assigned a .dwt extension and are stored in the designated , 131
Templates in AutoCAD allow you to easily start new drawings using setups , 129
**Test Bracket** , 155
**Test Bracket [INTERMEDIATE]** , 155
**Textures and Realistic Materials** , 4
The arc's radius , 43
The Array Creation Contextual Tab, 142
The ARRAYRECT command provides a robust alternative to basic copying, offering enhanced functionality , 141
The AutoCAD web and mobile apps , 3
the **Circle button** , 37
The current line width , 42
The dimensions on the drawing itself indicate the placement of these smaller figures , 126
the Dimensions panel , 159, 163, 164, 165, 167, 171, 172, 177, 205
the **Draw panel** , 27, 37, 38, 42, 102, 106, 107, 118, 194, 229
The Drawing Units Dialog Box, 103
**The Drawing Units dialog box uses drop-down lists, which is a commonly used feature in dialog boxes** , 103

The endpoint of the arc , 38
**The grid is currently being displayed within the limits of (0,0) and (12,9).** , 130
the Page setup settings , 308
**The Polar command will play a significant role in this scenario. Now, let's return to the identical dialog box** , 42
The Printer/Plotter Panel, 146
The radius is an important measurement in AutoCAD. , 38
**The symbol located on the right side of the edit box signifies the ability to generate arrays using mathematical relationships, expressions, and absolute values** , 143
This toolbar includes all the File commands mentioned in the Application Menu, as well as **Workspace** and **Undo/Redo options** , 8
To access the Files of type list, simply click on the arrow located to the right of the list box. By selecting this option, you will be able to access a comprehensive list of file types. , 132
To begin copying, simply right-click and choose the **Copy option**. AutoCAD will request the move point once more , 136
To close the drawing window, simply click on the Close button (X) located in the upper-right corner. Ensure that you close the current drawing without closing the entire AutoCAD application window , 134
To create a custom template in AutoCAD, simply design a drawing with your desired settings and save it as a template , 131
**To customize this toolbar, simply click the arrow at the end. This action will bring up the following options** , 8
To disable the grid from extending beyond the limits , 130
**To enable or disable Dynamic Input, simply click the corresponding button located on the Status bar** , 28
To execute the command, press the letter z followed by the **Enter key**. Then, press the letter a and hit Enter, or choose the "**All**" option from the command line , 129
To exit the grip editing system, simply press <**Enter**> or the spacebar , 136
To save your work, simply click on the **Save As tool** , 132
tolerance precision , 163
tolerances , 158, 161, 163, 202
total distance the rows will occupy , 85
tracking in AutoCAD , 42
**Tracking Settings** , 32, 42
**transformation in the cursor's appearance** , 57
transition from 2D drafting to 3D modeling , 2
triangle located next to the title. , 12
**Trim command** , 75, 82, 83
TRIM mode , 80
two lines are connected , 68
**two main components** , 11
**Two-place decimals are often preferred due to their widespread usage** , 105

## U

UCS icon , 8, 174

Under specific conditions, AutoCAD will open a ribbon tab that is not typically accessible. This tab is referred to as a contextual tab , 142
UNDERSTANDING DIMENSIONS, 158
Understanding Drawing Orientation, 148
Understanding Dynamic Input, 28
Understanding Dynamic Input in AutoCAD 2025, 28
Undo command, 25, 61, 300
Undo is a powerful command in AutoCAD , 25
undo multiple commands , 25
undo the last copy , 61
unique set of functions , 145
**Urban Planning** , 5
Use all polar tracking settings , 42
Use object snap to easily locate the quadrant point, 112
Use the <**Esc**> key to easily remove grips , 136, 140
Use the **array command** , 61
Use the Array Creation context tab , 85
use the **ARRAYRECT command** , 153
Use the Column panel , 85
Use the **Copy function** , 62, 63
**Use the copy tool to duplicate elements in your design** , 135
use the Perpendicular function , 33
Use the right-click function to complete the selection of objects. When using AutoCAD, you will be prompted to specify a base point or displacement , 135
Use the Rows panel , 85
useful commands , 68
Users will benefit from improved stability , 3
using ARRAY in AutoCAD , 151
Using Multifunctional Grips, 144
using OSNAP , 41, 50, 73
Using OSNAP Override, 35
using Polar Tracking in AutoCAD , 32
Using the Array Creation context tab , 85
Using the AutoCAD Web and Mobile Apps, 23
using the Circle command , 36
using the context tab , 89
using the Distance option to chamfer , 79
Using the Drafting Settings Dialog Box, 101
Using the File Tab, 24
Using the multiple copy function , 155
**using the one-by-one method** , 57
Using the Save As command, 22
Using the Save command, 21
Using the Undo and Redo commands, 25
using three decimal places , 124
using your Autodesk account credentials. , 23
utilizing the line tool , 68

utilizing two important commands , 69

## V

variable overall scale , 163
variety of text styles , 162
various components , 3
various essential functions , 16
various infrastructure projects , 4
**various locations** , 71, 154
Various plotters , 146
various units of measurement , 104
Verb/Noun Editing, 113
version of AutoCAD , 2, 3, 4, 6
View your recently opened files either as a list or as icons. , 6
ViewCube , 8
**visual representation** , 6, 28, 121, 138, 258
**visual representations** , 33
visualizations of 3D models. , 3

## W

***Weave* [INTERMEDIATE]** , 154
**welcome window** , 6
What are the Drafting Priorities, 27
What Is My Drawing Unit, 18
When using AutoCAD, it is possible to easily adjust the scale of your drawing to perfectly fit any desired drawing sheet size during the plotting process , 130
When you save a drawing, it can be transformed into a template , 131
Wide Range of Applications, 4
Wide Range of Applications in Various Industries, 4
wide range of industries , 5
wide range of powerful tools and features , 21
Window Mode (W), 54
Window Polygon Mode (WP), 55
working in AutoCAD , 54, 68, 82, 89, 129, 139, 158, 278
Workspace , 8, 117

## Y

You have adjusted the density of the drawing grid multiple times, consistently within the confines of a 12 × 9 space , 129
you will gain the expertise to modify the drawing grid by using the LIMITS command , 129

OceanofPDF.com